

HEAT TRANSFER PROGRAMS FOR THE DESIGN OF STRUCTURES EXPOSED TO FIRE

BY

J E Mason

Supervised by

Associate Professor Andrew H Buchanan

Fire Engineering Research Report 00/9

March 2000

This report was presented as a project report
as part of the M.E. (Fire) degree at the University of Canterbury

School of Engineering
University of Canterbury
Private Bag 4800
Christchurch, New Zealand

Phone 643 364-2250
Fax 643 364-2758

Acknowledgements

I would like to acknowledge the assistance and support I have received during the course of this project and during the time I have been studying for this degree.

My supervisor, Dr Andy Buchanan, for his suggestions for the subject of this report, and for his guidance and suggestions throughout this project, and also for his initiative in making this course available to part-time students.

Dr Jean-Marc Franssen of the University of Liège, for his patience and prompt response to my questions about the inner workings of SAFIR.

The New Zealand Fire Service for their financial support for the Master of Engineering in Fire Engineering course.

Sinclair Knight Merz for their financial support during this course.

Kaye for her patience and understanding during the four years I have been studying for this degree.

Abstract

The Finite Element Method was originally developed as a numerical method for stress analysis. It was very quickly applied to heat transfer analysis, and is now used for fluid mechanics, bio-mechanics, hydraulics, and many other problems too complex to be solved analytically. The recent rapid growth in the power of desktop computers has enabled the Finite Element Method to evolve from a tool that was difficult to use, expensive, and rarely used by most engineers to a tool that is now commonly used for analysis. The increasing power of computers can only widen the use of finite element analysis.

However, while finite element software is a powerful and indispensable tool, the programs are too often used without proper understanding of the underlying theory and of the limitations of the software.

SAFIR is a powerful finite element program for the thermal and structural analysis of structures exposed to fire. It is widely available for educational and research use. However, it is limited by the lack of a comprehensive Users' Manual, and the only Pre-processor, SAFIR Wizard, is limited to steel I sections.

This report reviews the use of the Finite Element Method for heat transfer analysis, and discusses some of the features and limitations of SAFIR with the aim of giving the user of SAFIR the understanding needed for reliable finite element modelling. A new Pre-processor has been developed, that widens the scope of the existing SAFIR Wizard for modelling steel sections, and adds Dycore and Hi-bond slabs.

Contents

ABSTRACT

CHAPTER 1 INTRODUCTION	1
1.1 Objective	1
1.2 Scope	2
1.3 SAFIR	2
1.4 Pre-Processing and Post-Processing for SAFIR	3
CHAPTER 2 SAFIR PRE-PROCESSOR	6
2.1 Introduction	6
2.2 Features	6
CHAPTER 3 THE FINITE ELEMENT METHOD IN HEAT TRANSFER ANALYSIS	9
3.1 Introduction	9
3.2 Background	9
3.3 General Formulation	10
3.4 Weighted Residual Methods	12
3.5 Shape Functions	13
3.6 Finite Element Formulation in Two Dimensions	16
3.7 Triangular (3 node) Elements	19
3.8 Rectangular (4 node) Element	25
3.9 Convective Boundary Conditions	28
3.10 Time-Dependant (Transient) Problems	29
3.11 Non-Linear Problems	33
CHAPTER 4 SAFIR	34
4.1 Introduction	34
4.2 Modelling Considerations	34
4.2.1 Finite Element Mesh	34

4.2.2	Bandwidth and the Effect of Renumbering	36
4.2.3	Renumbering in SAFIR	37
4.3	Materials	38
4.3.1	Standard Properties	38
4.3.2	Steel Materials	39
4.3.3	Concrete Materials	41
4.3.4	Gypsum	43
4.3.5	Accounting for Moisture Content in User-defined Materials	45
4.3.6	Specific Volume Enthalpy	46
4.4	Time Temperature Curves in SAFIR	47
4.5	Cavities	48
4.5.1	Convective Heat Transfer in Cavities	48
4.5.2	Radiative Heat Transfer in Cavities	49
CHAPTER 5	CONCLUSIONS AND RECOMMENDATIONS	53
5.1	Summary	53
5.2	Further Work	53
References		55
Appendices		

Appendix 1: SAFIR Pre-Processor Users' Manual

CHAPTER 1

INTRODUCTION

1.1 Objective

The objective of this study was to critically review the use of the finite element method for heat transfer analysis, leading to the production of a stand-alone heat transfer module.

The finite element method is a powerful and versatile tool for analysis. The advent of the desktop computer and the wide availability of commercial finite element software has made the use of the finite element method much more common in engineering practices. However, the successful use of these programs requires an understanding of the physical theories on which they are based, and of the limitations of the program. As stated by Cook et al (1989):

“Although the finite element method can make a good engineer better, it can make a poor engineer more dangerous”

SAFIR is a finite element program developed by Dr Jean-Marc Franssen at the University of Liège in Belgium. It was developed specifically for the analysis of structures at elevated temperatures. However, its useability is hampered by the lack of a comprehensive users' guide. A manual for SAFIR has been written (Nwosu et al 1999), but this is primarily a reference manual describing the format and preparation of the SAFIR data files.

This report presents a summary of the use of the finite element method for heat transfer analysis, and a discussion of some of the features and modelling characteristics of SAFIR, with the aim of providing users of SAFIR with some of the background information and guidance needed to have confidence in the validity of their models.

This report is not intended to be the definitive users' guide to SAFIR. It is intended to be read in conjunction with the SAFIR User Manual. Users new to finite

element analysis and SAFIR may find it useful to review this report first, and use the Nwosu manual as a reference when preparing the SAFIR data files.

1.2 Scope

This report is limited to discussion of the thermal part of SAFIR. The heat transfer part of SAFIR was of the most interest in the context of this research, and also the most relevant for the use of SAFIR in the Master of Fire Engineering degree at the University of Canterbury.

The Pre-processor developed during this study is limited to two-dimensional models. Developing and testing the program for two-dimensional models was a logical first step in the overall development. Many engineering thermal analysis problems are two-dimensional, so this initial restriction is not considered to be too limiting.

1.3 SAFIR

SAFIR is a special-purpose finite element programme for thermal analysis and/or structural analysis at elevated temperatures. SAFIR was developed at the University of Liège in Belgium, and is the second thermal analysis program to be developed at the University of Liège. An earlier programme, CEFICOSS (Computer Engineering of the Fire resistance of COMposite and Steel Structures) was also developed there (Franssen 1987 and Schleich 1987).

Two dimensional finite elements used are triangular (3 node) or quadrilateral (4 node). Only linear elements may be used for thermal analysis. Three-dimensional elements available include solid elements (six or eight nodes), shell, beam, and truss elements.

There are a number of standard materials available in the program (most based on Eurocode 2 and Eurocode 3), with thermal and mechanical properties for these materials generated automatically. In addition, each model can have two user-defined materials, with material properties defined by the user. The effect of moisture content on conductive heat transfer is modelled in the standard concrete materials and also in

one of the user-defined materials. Material properties can be non-linear and temperature dependent.

Two standard time-temperature distributions are built into the programme, ISO 834 and ASTM E119. User-defined time-temperature curves can also be specified, allowing linking to the results of compartment fire models (eg FAST) or the use of standard time-temperature distributions not included in the program (e.g. Eurocode parametric fires).

Within the section the program models heat transfer by conduction. Heat transfer by convection and radiation are modelled at the boundaries. Internal cavities are permitted in two-dimensional analyses (with some restrictions on geometry, see Chapter 4). Radiation and convection are modelled in internal cavities.

The SAFIR model is defined in a series of text files that can be created or edited with any standard text editor. The results of the analysis are presented in text files. Pre-processing and post-processing capability are available in separate programs.

1.4 Pre-Processing and Post-Processing for SAFIR

The input files for SAFIR can be created with a text editor but the process is tedious and prone to error, and there is no feasible means of checking and verifying the accuracy of the data. There are several programs available to facilitate the preparation of the input files for a two-dimensional thermal analysis and also to aid in the interpretation of the results of the analysis. There are currently no similar programs available to assist in the preparation of data files for a three-dimensional thermal analysis or a structural analysis.

SAFIR98 Wizard, developed at University of Liège, creates SAFIR data files for steel I-sections, with or without profile insulation. In a sequence of steps, the user is prompted for parameters required to define the model. At the end of the sequence, a set of SAFIR data files are written to disk. The programme includes European and American sections, and was modified by the author to add BHP Universal Beam and Universal Column sections. The program is limited to the creation of data files for these sections. It cannot open existing data files to allow modification.

Because of the limited scope of the SAFIR Wizard, and the lack of any other program to assist in the creation and checking of SAFIR data files, it was decided to develop a fully interactive program to aid in the creation and modification of data files, and to facilitate checking the input. SAFIR Pre-processor was written by the author (see appendix A). It is an interactive program for the creation and modification of SAFIR data files for two-dimensional thermal analysis. A brief description of the capabilities of the Pre-Processor is given in Chapter 2.

DIAMOND97 is the SAFIR post-processor developed at the University of Liège. It enables the plotting of temperature contours from a two-dimensional analysis. The program can also animate the temperature contours dynamically to show how the temperature varies with time. A graph can be drawn for any node, showing how the temperature varies with time.

SAFIR2XL is a small utility, also developed by the author. It converts the output file of a SAFIR analysis into a comma-delimited file that can be read into Excel, or any other spreadsheet program. The nodal temperatures can then be plotted or further processed. This may give more flexibility for graphical interpretation of the results than is available in DIAMOND97.

The relationship of the various programs in a SAFIR analysis is shown in Figure 1.1.

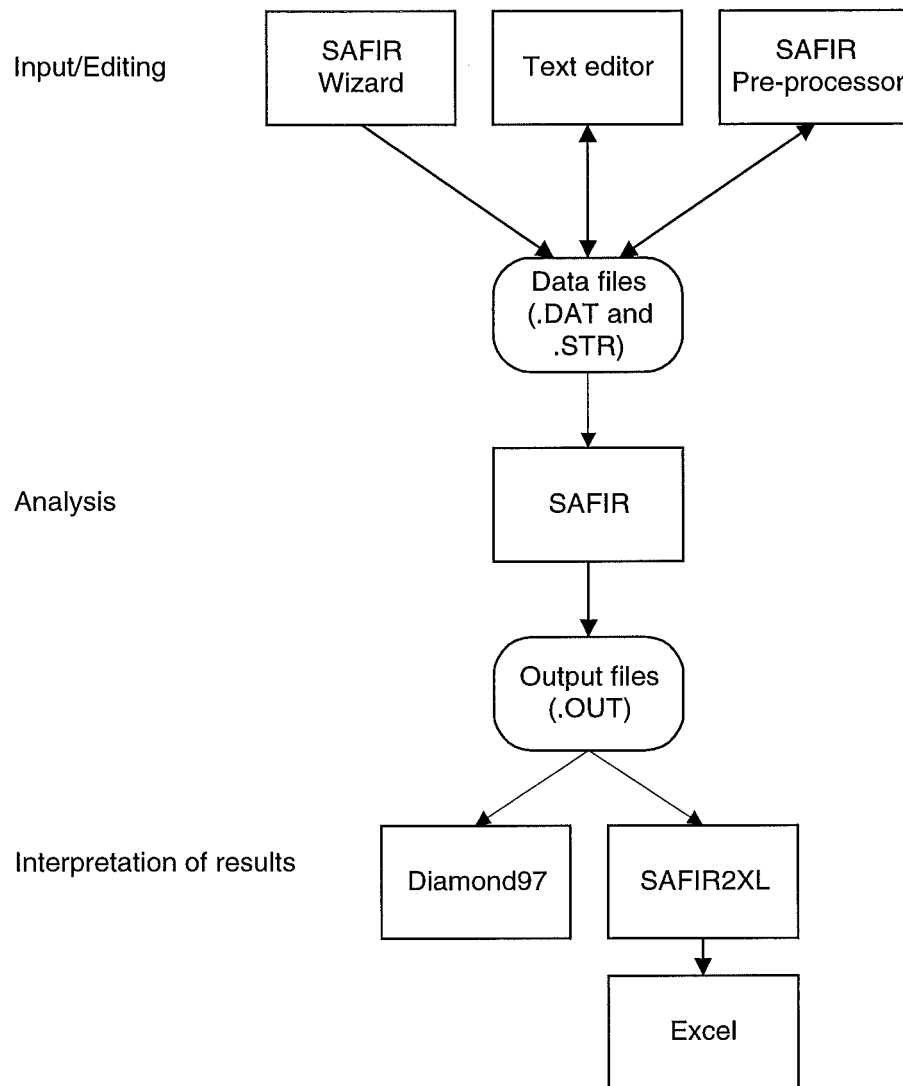


Figure 1.1. Relationship of programs in a SAFIR thermal analysis.

CHAPTER 2

SAFIR PRE-PROCESSOR

2.1 Introduction

The existing SAFIR Wizard has a limited scope. It can create data files for steel I-sections, with or without profile insulation, for two-dimensional thermal analyses. However it does not allow the user to modify existing SAFIR data files. SAFIR is a powerful program for thermal analysis, but the need to use a text editor to prepare data files for models other than those available in the Wizard was a significant hindrance to the use of the program. It was decided to develop a fully interactive graphical program for the creation and modification of SAFIR data files. SAFIR Pre-Processor was developed by the author to this end. This chapter describes the features of the Pre-Processor. A full Users' Manual can be found in Appendix 1.

2.2 Features

SAFIR Pre-Processor is an interactive program for the creation and modification of SAFIR data files for two-dimensional thermal analysis. Standard sections with a pre-defined mesh facilitate the creation of new SAFIR models. The standard sections available include steel I-sections, hollow core concrete slabs (Dycore), and profiled metal deck composite concrete slabs (Hi-Bond). Steel section libraries available include Australasian, American, and European I sections. These sections can have profile insulation or box insulation added. Dycore and Hi-Bond models include varying slab thicknesses.

A basic set of CAD-like tools allow the user to interactively manipulate the model, enabling refinement of the mesh, or allowing the user to quickly modify the standard models. For example, a steel I-section can be easily modified to a model of a concrete encased steel column. When modifying the finite element mesh, the user can delete elements, sub-divide elements to give a finer mesh, merge two or more elements into one, and create new elements by copying existing elements or drawing new elements on the screen.

Any existing two-dimensional thermal SAFIR data file can be opened in the Pro-Processor and further manipulated. The Pre-processor can be used interchangeably with the other input methods, allowing the user to choose the method that is most convenient for the particular input task. For example, minor modifications to the data file may be made more conveniently with a text editor. Doing so does not prevent the user from subsequently editing the same file with the Pre-Processor.

The pre-processor allows the user to apply the ISO 834 or ASTM E119 time-temperature curves to any selected element. SAFIR also supports user-defined time temperature curve. The pre-processor takes advantage of this by automatically generating time-temperature data for ISO 834 fires with a linear decay phase and Eurocode parametric fires calculated from parameters provided by the user.

All materials supported by SAFIR are available, including the user-defined material. Material properties can be applied to any selected element. User-defined material properties are supported by the Pre-Processor, and these properties can be plotted on the screen as they are entered to facilitate confirmation of the data.

CAD-like viewing features allow the user to zoom into the model. Node and element numbers can be plotted. Printing is supported, so that the screen view can be echoed to a printer.

The main window of the Pre-Processor is shown in Figure 2.1.

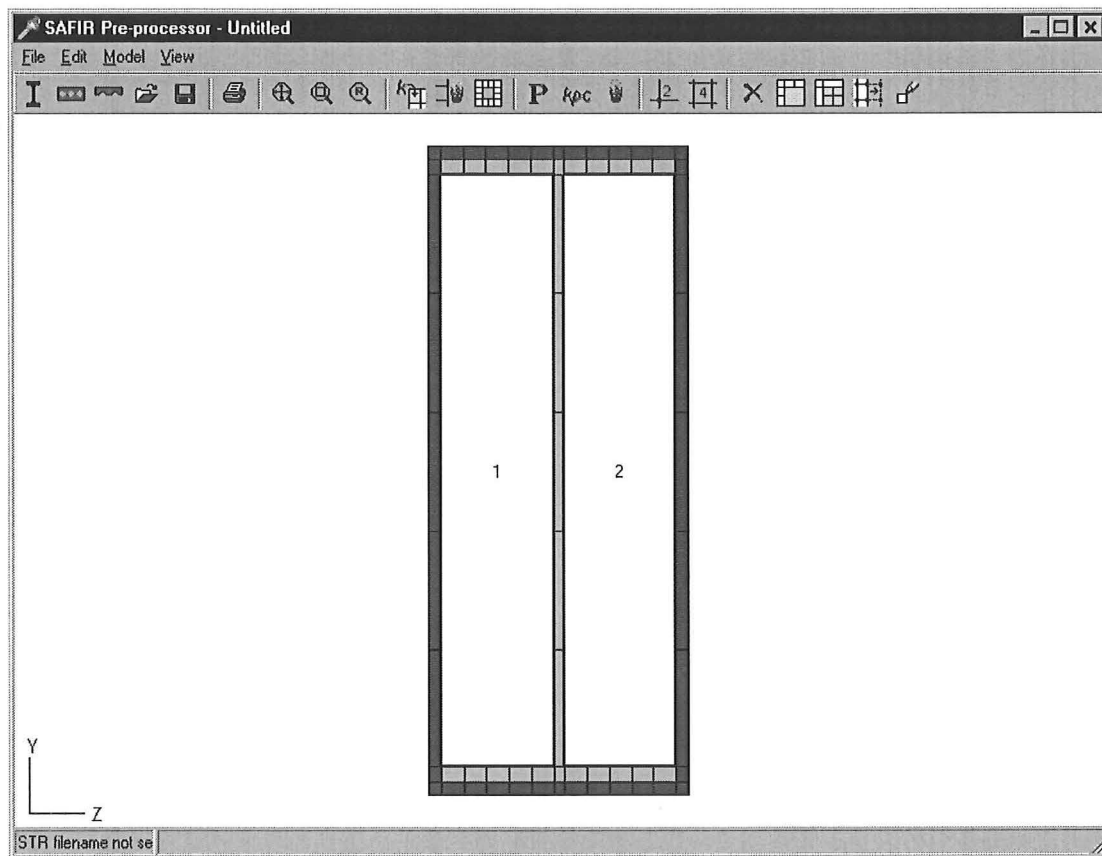


Figure 2.1. SAFIR Pre-Processor main window.

CHAPTER 3

THE FINITE ELEMENT METHOD IN HEAT TRANSFER ANALYSIS

3.1 Introduction

This chapter reviews the finite element method for heat transfer analysis, describing the principles of the finite element method and the derivation of the matrix form of the heat transfer equations. The derivation of the element stiffness matrices for two simple element types used in SAFIR is explained. The chapter concludes with a brief discussion of non-linear and time-dependant finite element analysis. Only two-dimensional analysis is discussed here, three-dimensional analysis is a relatively straight forward extension of the theory for two dimensions.

The material in this chapter results from a review of a number of references, most notably Lewis et al (1996), Reddy & Gartling (1994), and Zienkiewicz & Taylor (1998).

3.2 Background

The Finite Element Method is a numerical method for analysing continuous domains. Exact solutions of the governing equations for most practical problems can only be obtained for simple problems or problems in which restrictive assumptions are made with respect to geometry, material properties, and/or boundary conditions. Numerical solution methods are needed for the analysis of practical problems. The finite element method is commonly used for problems that are too complicated to be solved by classical analytical methods.

The Finite Element Method views a domain as an assembly of simple geometric shapes (called finite elements). The method is based on the concept that the solution of a differential equation can be reformulated as a linear combination of a series of unknown parameters and appropriately selected functions (called *approximation functions* or *interpolation functions*). The approximation functions are selected so that they satisfy the boundary conditions. Real problems are often based

on geometric complex regions, and it is difficult to select approximation functions that satisfy the boundary conditions. If the region can be represented as a series of simple subdomains (or finite elements) that permit of the generation of approximation functions that satisfy the boundary conditions of each element, then the problem should be able to be solved.

The finite element method as it is known today was first proposed by Courant in a paper published in 1943 (Cook et al 1989). By the early 1950's engineers had written the stiffness equation in matrix form and solved the equations on digital computers. At this time a model with 100 degrees of freedom was considered to be a large problem.

Most of the early development in the Finite Element Method was done in the aerospace industry. Widespread use of the finite element method began in 1953 when Turner (Boeing Airplane Company) suggested that triangular plane stress elements could be used to model the skin of a delta wing.

The term "Finite Element Method" was coined by Clough in 1960 (Cook et al 1989). At this stage the formulations of different element types were based largely on intuition, and was not until 1963 that a mathematical foundation for Finite Element Analysis was developed. This sparked the general acceptance of the Finite Element Method.

Papers about the application of the finite element method to heat conduction and seepage flow started to appear in the late 1960's. General-purpose finite element computer programs appeared during the late 1960's and early 1970's. The proliferation of desktop computers in the 1980's, and particularly the more powerful computers available in the 1990's, has made the Finite Element Method available to all engineers.

3.3 General Formulation

The general equation for two-dimensional heat conduction in an isotropic material is

$$\left(k \frac{\partial^2 T}{\partial x^2} + k \frac{\partial^2 T}{\partial y^2} \right) + Q = \rho c \frac{\partial T}{\partial t} \quad (3.1a)$$

or

$$k \nabla^2 T + Q = \rho c \frac{\partial T}{\partial t} \quad (3.1b)$$

where k is the thermal conductivity of the material, T is the temperature, Q is the amount heat generated in the material per unit volume, ρ is the density, c is the heat capacity, and t is time.

If steady-state conditions exist, i.e. the temperature does not change with time, then the equation reduces to

$$k \nabla^2 T + Q = 0 \quad (3.2)$$

If there is no internal heat generation then the equation further reduces to

$$k \nabla^2 T = 0 \quad (3.3)$$

The boundary conditions may be a known temperature and/or a known heat flux (both conditions may not be specified over the same part of the boundary).

Known temperature

The temperature may be specified, i.e. $T = T_0$

Known Heat flux

The heat flow may be specified. This requires that

$$q = -k \frac{\partial T}{\partial n} \quad (3.4a)$$

or

$$k \frac{\partial T}{\partial n} + q = 0 \quad (3.4b)$$

where q is the specified heat flow, and n is the outward normal vector to the boundary.

This can be written in the alternative form

$$k \left(\frac{\partial T}{\partial x} n_x + \frac{\partial T}{\partial y} n_y \right) + q = 0 \quad (3.4c)$$

where n_x and n_y are the components of the outward normal vector parallel to the x and y axes.

For the heat transfer problem to be defined, at least one of these conditions must be specified on at least part of the boundary.

This classical formulation (equation 3.1b) is suitable for solution by the finite difference method, but must be replaced by a variational formulation for Finite Element Analysis. A weighted residual method is normally used for this.

3.4 Weighted Residual Methods

Several methods can be used to transform the heat transfer equation to a form suitable for Finite Element Analysis. The most common is the Weighted Residual Method.

Consider a general differential equation of the form

$$a \frac{\partial^2 u}{\partial x^2} + b = 0 \quad (3.5)$$

The solution of this equation, u , can be approximated by a function

$$\bar{u}(x) = a_1 \phi_1(x) + a_2 \phi_2(x) + \dots + a_n \phi_n(x)$$

or

$$\bar{u}(x) = \sum_{i=1}^n a_i \phi_i(x) \quad (3.6)$$

\bar{u} is an approximation for u , so when (3.6) is substituted into (3.5), the equation will not necessarily equate to zero, i.e.

$$a \frac{\partial^2 \bar{u}}{\partial x^2} + b = e(x) \quad (3.7)$$

where $e(x)$ is a non-zero residual.

The weighted residual method requires that the weighted averages of the residual be equal to zero, so

$$\int_{x_1}^{x_2} w_i(x) e(x) dx = 0 \quad (3.8)$$

where $w_i(x)$ is a set of weighting functions.

Theoretically, any set of weighting functions could be selected, and there are several methods used in finite element formulation that use different weighting functions. The most common weighted residual method is the Galerkin method, which equates the weighting function to global *shape functions*. The advantage of the Galerkin method over the other weighted residual methods is that it usually results in symmetrical matrices.

3.5 Shape Functions

Consider a one-dimensional heat transfer problem, e.g. radial heat transfer in a long cylinder, subjected to a heat flux of q /unit length in the inner surface, and with a fixed temperature of T_o at the outer surface. The exact solution, when the internal heat generation is zero, is

$$T(r) = T_o - \frac{q}{k} \ln \left(\frac{r}{R_2} \right) \quad (3.9)$$

where R_2 is the radius to the outer surface, and k is the thermal conductivity.

Consider a 2 node one-dimensional element with node 1 on the outside face and node 2 on the inside face of the cylinder (Figure 3.1). Approximate the actual temperature distribution with a linear temperature distribution. The temperature distribution can be given in terms of two linear functions, N_1 and N_2 , each of which

take the value of unity at the node with which they are associated and are zero at the other node.

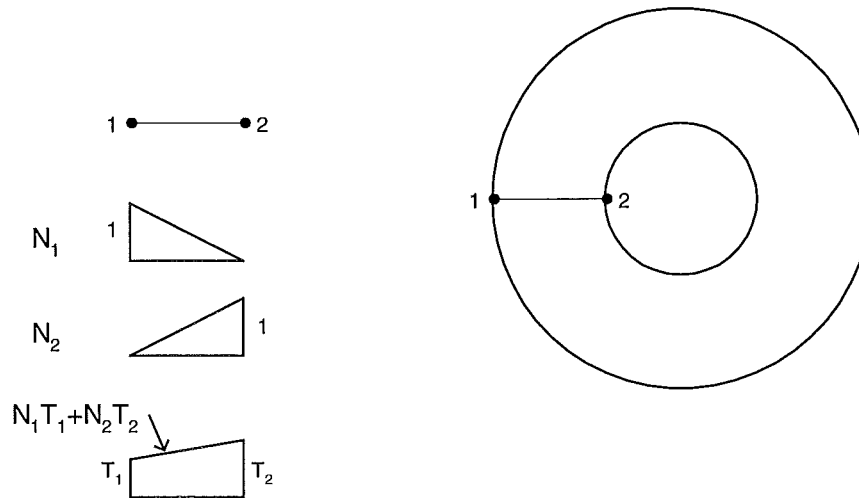


Figure 3.1 One element representation of a one-dimensional finite element model (Owen & Hinton 1980).

A more accurate solution would be found if the radius were divided into three elements and four nodes (Figure 3.2). Within each element the temperature varies linearly.

Consider a set of approximation functions $N_1(r)$, $N_2(r)$, $N_3(r)$, and $N_4(r)$ associated with each node in the element subdivision. A typical approximation function $N_i(r)$ is zero over the whole mesh, except in the elements connected to node i .

At node i the function has a value of unity, within the element associated with node i the function is linear. These functions are called the *shape functions*.

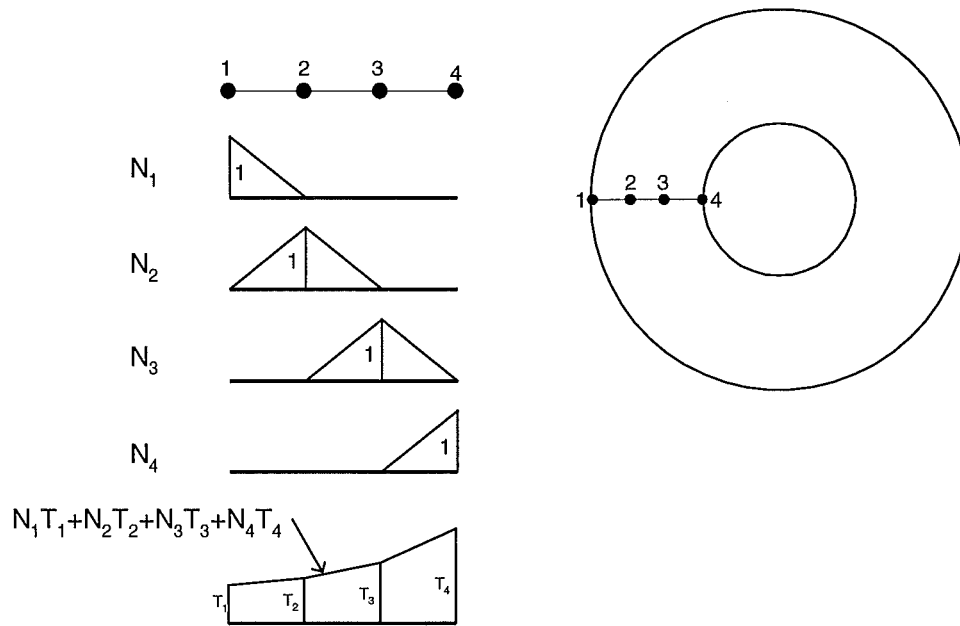


Figure 3.2 Three element representation of a one-dimensional finite element model (Owen & Hinton 1980).

It can be seen that the temperature distribution can be represented by the expression

$$T(r) = N_1 T_1 + N_2 T_2 + N_3 T_3 + N_4 T_4 \quad (3.10)$$

For an arbitrary number of elements this can be expressed as

$$T(r) = \sum_{i=1}^n N_i T_i \quad (3.11)$$

where N_i are the shape functions, and T_i are the nodal temperatures.

Note the similarity of equation (3.11) with equation (3.6).

In the Galerkin form of the weighted residual method, the weighting functions correspond to the shape functions, i.e. $w_i = N_i$ of equation 3.11, or $w_i = \phi_i$ of equation 3.6.

3.6 Finite Element Formulation in Two Dimensions

Recall the general form of the two dimensional steady state heat transfer equation (equation 3.2)

$$k\nabla^2 T + Q = 0 \quad (3.12)$$

or

$$\left(k \frac{\partial^2 T}{\partial x^2} + k \frac{\partial^2 T}{\partial y^2} \right) + Q = 0 \quad (3.13)$$

The method of weighted residuals requires that the weighted averages of the residual be equal to zero

$$\int_A w_i e dA = 0 \quad (3.14)$$

Now, if T is an approximation to the exact temperature distribution, then we can set

$$e = k \frac{\partial^2 T}{\partial x^2} + k \frac{\partial^2 T}{\partial y^2} + Q \quad (3.15)$$

Substituting equation 3.15 into equation 3.14 gives

$$\int_A w_i \left[k \frac{\partial^2 T}{\partial x^2} + k \frac{\partial^2 T}{\partial y^2} + Q \right] dA = 0 \quad (3.16)$$

Green's Theorem (Zienkiewicz 1989) states that the integrals of two arbitrary functions ϕ and φ over an area A and its boundary S are related as follows

$$\int_A \left(\frac{\partial \phi}{\partial x} \frac{\partial \varphi}{\partial x} + \phi \frac{\partial^2 \varphi}{\partial x^2} \right) dA = \int_S \phi \frac{\partial \varphi}{\partial x} n_x dS$$

$$\int_A \left(\frac{\partial \phi}{\partial y} \frac{\partial \varphi}{\partial y} + \phi \frac{\partial^2 \varphi}{\partial y^2} \right) dA = \int_S \phi \frac{\partial \varphi}{\partial y} n_y dS$$

Applying Green's Theorem to equation 3.16 results in

$$\int_S \left[k \frac{\partial T}{\partial x} n_x + k \frac{\partial T}{\partial y} n_y \right] w dS - \int_A \left[k \frac{\partial w}{\partial x} \frac{\partial T}{\partial x} + k \frac{\partial w}{\partial y} \frac{\partial T}{\partial y} - Qw \right] dA = 0 \quad (3.17)$$

Now consider the boundary conditions: the temperature specified over part of the boundary, and the heat flux specified over part of the boundary. At points where the temperature is specified the value of the weighting function is zero. Therefore the first integral in equation 3.17 becomes zero where the temperature is specified. On those parts of the boundary where the heat flux is specified (from equation 3.4c)

$$k \left(\frac{\partial T}{\partial x} n_x + \frac{\partial T}{\partial y} n_y \right) + q = 0$$

or

$$k \left(\frac{\partial T}{\partial x} n_x + \frac{\partial T}{\partial y} n_y \right) = -q \quad (3.18)$$

Substituting equation 3.18 into equation 3.17 gives

$$\int_S q w dS + \int_A \left[k \frac{\partial w}{\partial x} \frac{\partial T}{\partial x} + k \frac{\partial w}{\partial y} \frac{\partial T}{\partial y} - Qw \right] dA = 0 \quad (3.19)$$

Recall that the temperature field can be approximated as

$$T = \sum_{i=1}^n N_i T_i \quad (3.20)$$

where N_i are the global shape functions and T_i are the unknown nodal temperatures.

Also note that Galerkin's method sets the weighting function equal to the shape functions

$$w_j = N_j \quad (3.21)$$

Equation 3.20 and 3.21 are substituted into equation 3.19 to give

$$\int_S q N_j dS + \int_A \left[k \sum_{i=1}^n \frac{\partial N_j}{\partial x} \frac{\partial N_i}{\partial x} T_i + k \sum_{i=1}^n \frac{\partial N_j}{\partial y} \frac{\partial N_i}{\partial y} T_i - Q N_j \right] dA = 0$$

This can be rearranged to give

$$\int_A \left[k \sum_{i=1}^n \frac{\partial N_j}{\partial x} \frac{\partial N_i}{\partial x} T_i + k \sum_{i=1}^n \frac{\partial N_j}{\partial y} \frac{\partial N_i}{\partial y} T_i \right] dA = \int_A Q N_j dA - \int_S q N_j dS$$

The order of summation and integration can be interchanged

$$\sum_{i=1}^n \left[\int_A k \left(\frac{\partial N_i}{\partial x} \frac{\partial N_j}{\partial x} + \frac{\partial N_i}{\partial y} \frac{\partial N_j}{\partial y} \right) dA \right] T_i = \int_A Q N_j dA - \int_S q N_j dS$$

This represents a series of n equations, which may be written in matrix form as:

$$[K]\{a\}=\{f\}$$

[K] is the global “stiffness” matrix, also called the conductivity matrix. {f} is the “force vector”. The components of these matrices are:

$$K_{ij} = \int_A k \left[\frac{\partial N_i}{\partial x} \frac{\partial N_j}{\partial x} + \frac{\partial N_i}{\partial y} \frac{\partial N_j}{\partial y} \right] dA \quad (3.22)$$

$$f_i = \int_A Q N_i dA - \int_S q N_i dS \quad (3.23)$$

In practice, these matrices are established for each element separately, and then assembled to give the global matrices (Owen and Hinton 1980). The global matrices can then be solved for the nodal temperatures by any numerical solution technique.

To show how the element conductivity matrices are derived in practice, the matrices for two types of elements used in SAFIR will be derived in the following section.

3.7 Triangular (3 node) Elements

Consider a triangular element, shown in Figure 3.1. The element has three nodes. Each node has one degree of freedom, corresponding to the temperature at the node.

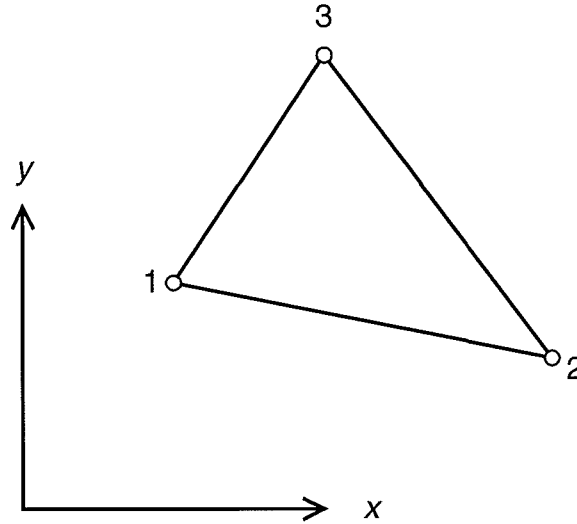


Figure 3.1. Triangular finite element.

As the temperature distribution within the element is linear, an expression for the temperature at any point within the element in terms of the x and y coordinates of that point can be written:

$$T = a + bx + cy \quad (3.24)$$

where a , b , and c are constants. If the known temperatures at the nodes are T_1 , T_2 , and T_3 then the values of the constants a , b , and c can be found by substituting the known nodal temperatures into equation 3.24:

$$\begin{aligned} T_1 &= a + bx_1 + cy_1 \\ T_2 &= a + bx_2 + cy_2 \\ T_3 &= a + bx_3 + cy_3 \end{aligned} \quad (3.25)$$

where x_i and y_i are the coordinates of node i .

Solving equation 3.25 for a , b , and c and substituting into equation 3.24 gives

$$T = \frac{1}{2A} [(\alpha_1 + \beta_1 x + \gamma_1 y)T_1 + (\alpha_2 + \beta_2 x + \gamma_2 y)T_2 + (\alpha_3 + \beta_3 x + \gamma_3 y)T_3] \quad (3.26)$$

where A is the area of the element, given by

$$A = \frac{1}{2} \text{Det} \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix}$$

or

$$A = \frac{1}{2} (x_1 y_2 + x_2 y_3 + x_3 y_1 - x_1 y_3 - x_2 y_1 - x_3 y_2)$$

and

$$\alpha_1 = x_2 y_3 - x_3 y_2$$

$$\beta_1 = y_2 - y_3$$

$$\gamma_1 = x_3 - x_2$$

with the other components given by cyclic permutation of the subscripts.

The shape functions can be written as

$$N_i = \frac{1}{2A} (\alpha_i + \beta_i x + \gamma_i y) \quad (3.27)$$

This function is linear, and can be shown to have a value of unity at the node to which it relates and zero at the other nodes. These are the criteria required for the shape function. The profile of a shape function is shown in Figure 3.2.

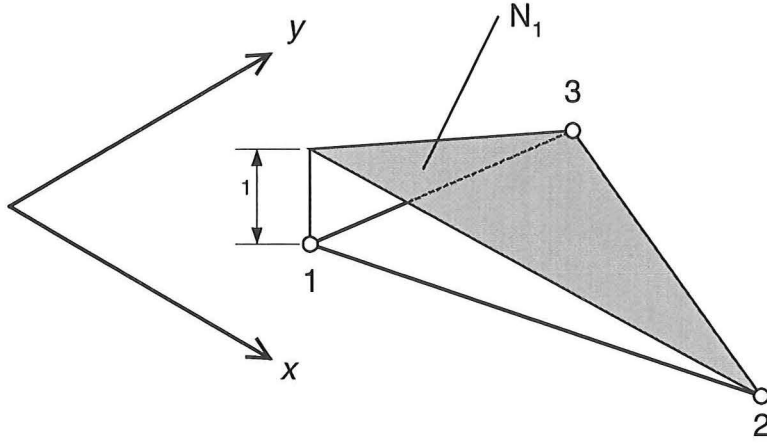


Figure 3.2. Typical shape function for a triangular element (Owen & Hinton 1980).

Equation 3.26 can then be written as

$$T = \sum_{i=1}^3 N_i T_i \quad (3.28)$$

The stiffness matrix for the triangular element can be found by substituting equation 3.27 into equation 3.22, giving

$$K_{ij} = \int_A \frac{k}{4A^2} (\beta_i \beta_j + \gamma_i \gamma_j) dA$$

All the terms inside the integral are constant, so we obtain

$$K_{ij} = \frac{k}{4A} (\beta_i \beta_j + \gamma_i \gamma_j)$$

The element stiffness matrix may be written as

$$\mathbf{K} = \frac{k}{4A} \begin{bmatrix} \beta_1 \beta_1 + \gamma_1 \gamma_1 & \beta_2 \beta_1 + \gamma_2 \gamma_1 & \beta_3 \beta_1 + \gamma_3 \gamma_1 \\ \beta_1 \beta_2 + \gamma_1 \gamma_2 & \beta_2 \beta_2 + \gamma_2 \gamma_2 & \beta_3 \beta_2 + \gamma_3 \gamma_2 \\ \beta_1 \beta_3 + \gamma_1 \gamma_3 & \beta_2 \beta_3 + \gamma_2 \gamma_3 & \beta_3 \beta_3 + \gamma_3 \gamma_3 \end{bmatrix} \quad (3.29)$$

where

$$\beta_1 = y_2 - y_3$$

$$\gamma_1 = x_3 - x_2$$

with the other components given by cyclic permutation of the subscripts. Notice that the stiffness matrix is symmetric.

The force vector is given by equation 3.23. The integration required by this equation is simplified by the use of *area coordinates*. Figure 3.3 shows a triangular element, with a point P at an arbitrary location within the element.

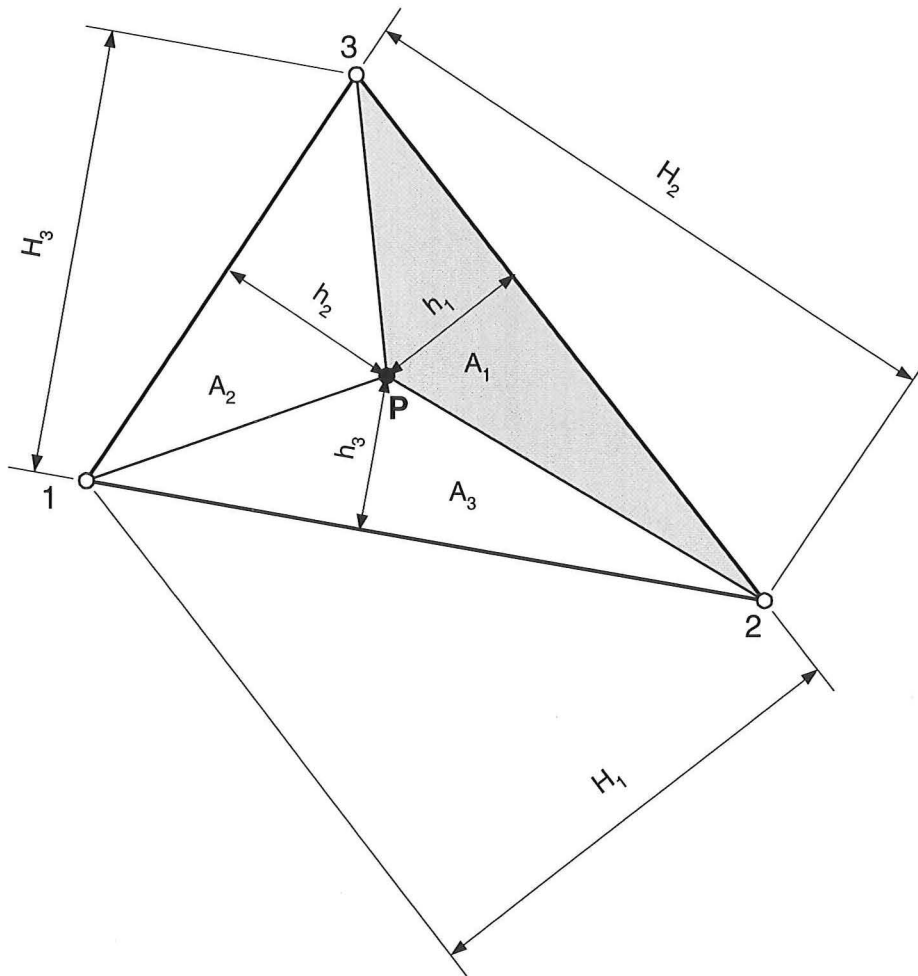


Figure 3.3. Area coordinates.

The location of P can be expressed in terms of dimensionless terms L_1 , L_2 , L_3 . Define L_1 as

$$L_1 = \frac{h_1}{H_1}$$

Similarly

$$L_2 = \frac{h_2}{H_2}, \quad L_3 = \frac{h_3}{H_3}$$

Notice that the area of the triangle is

$$A = \frac{BH_1}{2}$$

and the area of the sub-triangle A_1 is

$$A_1 = \frac{Bh_1}{2}$$

Clearly

$$L_1 = \frac{A_1}{A}$$

and generally

$$L_i = \frac{A_i}{A}$$

The coordinates L_i are termed area coordinates because they are defined by the areas of the sub-triangles.

It is obvious from Figure 3.3 that L_1 has the value of unity at node 1 and is zero at the other two nodes. This is exactly the distribution and values required for the shape functions. So the shape functions for a triangular finite element are equal to the area coordinates. There are two useful formulae that can be used for the integration over areas and surfaces (Owen and Hinton 1980):

$$\int_A L_1^a L_2^b L_3^c dA = \frac{a!b!c!}{(a+b+c+2)!} 2A \quad (3.30)$$

$$\int_S L_1^a L_2^b dS = \frac{a!b!}{(a+b+1)!} S \quad (3.31)$$

where a , b , and c are integer powers.

The first term in equation 3.23 can be written as

$$\{f\}_Q = Q \int_A \begin{bmatrix} N_1 \\ N_2 \\ N_3 \end{bmatrix} dA \quad (3.32)$$

Replacing the shape functions with the area coordinates and using equation 3.30 results in

$$\{f\}_Q = \begin{bmatrix} QA/3 \\ QA/3 \\ QA/3 \end{bmatrix} \quad (3.33)$$

The second integral of equation 3.23 depends on which side of the triangular element is subjected to the boundary condition. If side 1-2 is subject to the boundary condition then

$$\{f\}_{q,12} = - \int_{S_{12}} q N_i dS = -q \int_{S_{12}} \begin{bmatrix} N_1 \\ N_2 \\ 0 \end{bmatrix} dS \quad (3.34)$$

where S_{12} denotes integration along side 1-2 of the element. The zero term in the matrix arises because N_3 is zero along side 1-2. Substituting L_i for N_i in equation 3.34 and using equation 3.31 results in

$$\{f\}_{q,12} = \frac{-qS_{12}}{2} \begin{bmatrix} 1 \\ 1 \\ 0 \end{bmatrix} \quad (3.35)$$

Similarly

$$\{f\}_{q,23} = \frac{-qS_{23}}{2} \begin{bmatrix} 0 \\ 1 \\ 1 \end{bmatrix}$$

$$\{f\}_{q,31} = \frac{-qS_{31}}{2} \begin{bmatrix} 1 \\ 0 \\ 1 \end{bmatrix}$$

Then the nodal force vector is

$$\{f\} = \{f\}_Q + \{f\}_{q,12} + \{f\}_{q,23} + \{f\}_{q,31}$$

3.8 Rectangular (4 node) Element

Consider a rectangular element with 4 nodes (Figure 3.1). In addition to the global coordinate system x,y , there is a normalised coordinate system ξ,η with its origin at the centre of the element.

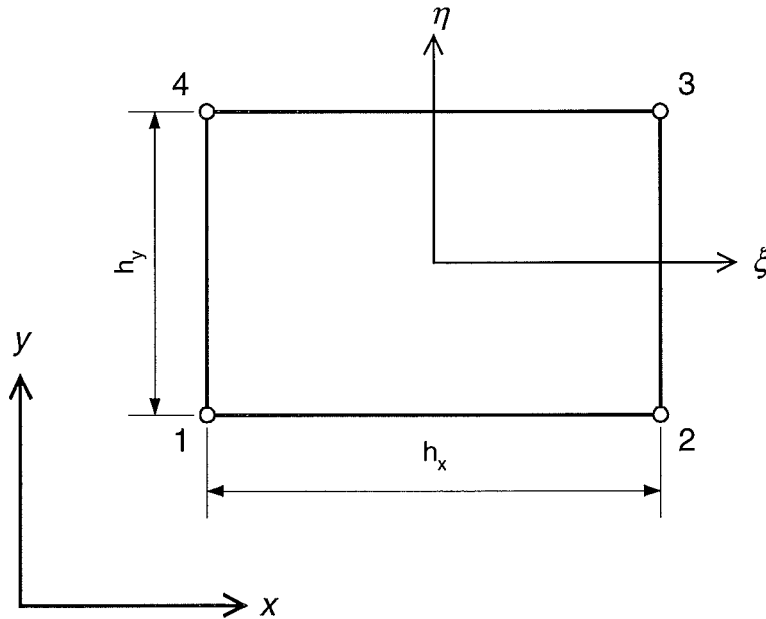


Figure 3.1. Rectangular element and normalised coordinate system (Reddy & Gartling 1994).

The normalised coordinates are related to the global coordinates by:

$$\xi = 2 \frac{x - x_c}{h_x} \quad (3.36)$$

$$\eta = 2 \frac{y - y_c}{h_y} \quad (3.37)$$

where x_c and y_c are the coordinates of the centre of the element, and ξ and η are dimensionless coordinates varying from -1 to $+1$.

The shape functions for this element are written in terms of the normalised coordinates:

$$\begin{aligned} N_1 &= (1 - \xi)(1 - \eta)/4 \\ N_2 &= (1 + \xi)(1 - \eta)/4 \\ N_3 &= (1 + \xi)(1 + \eta)/4 \\ N_4 &= (1 - \xi)(1 + \eta)/4 \end{aligned} \quad (3.38)$$

A shape function for the rectangular element is shown in Figure 3.2.

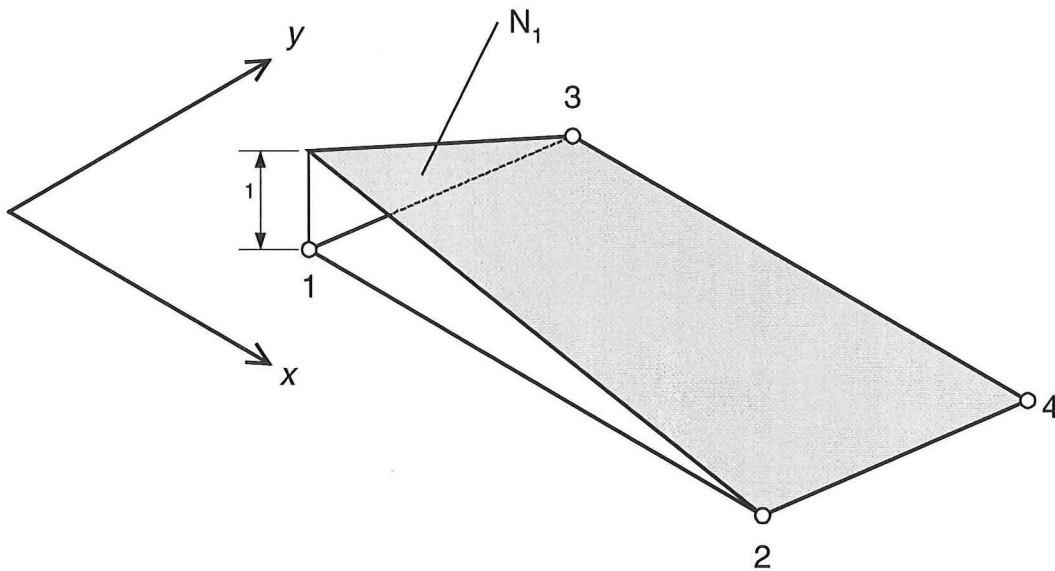


Figure 3.2. A typical rectangular element shape function.

The x,y coordinate system may be related to the ξ,η coordinate system by the following relationship:

$$\begin{aligned} x &= \sum_{i=1}^4 x_i N_i \\ y &= \sum_{i=1}^4 y_i N_i \end{aligned} \quad (3.39)$$

Now, from equation 3.36 it is evident that

$$x = \frac{1}{2} \xi h_x + x_c$$

and therefore

$$dx = \frac{1}{2} h_x d\xi \quad (3.40)$$

Similarly

$$dy = \frac{1}{2} h_y d\eta \quad (3.41)$$

The components of the stiffness matrix are given by (equation 3.22)

$$K_{ij} = \int_A k \left[\frac{\partial N_i}{\partial x} \frac{\partial N_j}{\partial x} + \frac{\partial N_i}{\partial y} \frac{\partial N_j}{\partial y} \right] dA \quad (3.42)$$

Substituting 3.40 and 3.41 into 3.42 we obtain

$$K_{ij} = \frac{1}{h_x h_y} \int_{-1}^1 \int_{-1}^1 k \left(h_y^2 \frac{\partial N_i}{\partial \xi} \frac{\partial N_j}{\partial \xi} + h_x^2 \frac{\partial N_i}{\partial \eta} \frac{\partial N_j}{\partial \eta} \right) d\xi d\eta \quad (3.43)$$

Substituting the shape functions from equation 3.38 into equation 3.43 gives the conductivity matrix for the rectangular element

$$[K] = \frac{k}{3h_x h_y} \begin{bmatrix} h_x^2 + h_y^2 & h_x^2/2 - h_y^2 & -(h_x^2 + h_y^2)/2 & -h_x^2 + h_y^2/2 \\ h_x^2/2 - h_y^2 & h_x^2 + h_y^2 & -h_x^2 + h_y^2/2 & -(h_x^2 + h_y^2)/2 \\ -(h_x^2 + h_y^2)/2 & -h_x^2 + h_y^2/2 & h_x^2 + h_y^2 & h_x^2/2 - h_y^2 \\ -h_x^2 + h_y^2/2 & -(h_x^2 + h_y^2)/2 & h_x^2/2 - h_y^2 & h_x^2 + h_y^2 \end{bmatrix} \quad (3.44)$$

The force vector was given in equation 3.23

$$f_i = \int_A Q N_i dA - \int_S q N_i dS \quad (3.45)$$

If the first term in this equation is replaced with

$$\int_{-1}^1 \int_{-1}^1 Q N_i d\xi d\eta$$

and if the shape function equations are substituted then

$$\int_A Q N_i dA = \begin{bmatrix} h_x h_y / 4 \\ h_x h_y / 4 \\ h_x h_y / 4 \\ h_x h_y / 4 \end{bmatrix}$$

The second term of equation is obtained in the same way as for the triangular element, except that area coordinates are not needed

$$\{f\}_{q,12} = \frac{-qS_{12}}{2} \begin{bmatrix} 1 \\ 1 \\ 0 \\ 0 \end{bmatrix}$$

The expression for the other three sides is similar.

3.9 Convective Boundary Conditions

The derivation of the finite element formulation in Section 3.6 ignored convective heat transfer at the boundaries. To account for convection requires a

modification of the basic formulation already developed. Convection is described quantitatively by Newton's Law of Cooling:

$$q = h(T_s - T_\infty)$$

where q is the convective heat transfer rate per unit area of surface, h is the heat transfer coefficient, T_s is the surface temperature of the body, and T_∞ is the ambient temperature of surrounding medium.

Equation 3.18 then becomes

$$k \left(\frac{\partial T}{\partial x} n_x + \frac{\partial T}{\partial y} n_y \right) = -(q + h(T - T_\infty))$$

Substituting this into equation 3.17 yields

$$\int_S q w + h(T - T_\infty) dS + \int_A \left[k \frac{\partial w}{\partial x} \frac{\partial T}{\partial x} + k \frac{\partial w}{\partial y} \frac{\partial T}{\partial y} - Q w \right] dA = 0$$

Using equations 3.20 and 3.21 and rearranging gives

$$\sum_{i=1}^n \left[\int_A k \left(\frac{\partial N_i}{\partial x} \frac{\partial N_j}{\partial x} + \frac{\partial N_i}{\partial y} \frac{\partial N_j}{\partial y} \right) dA \right] T_i + \sum_{i=1}^n \left[\int_S h N_i N_j dS \right] T_i = \int_A Q N_j dA - \int_S q N_j dS + \int_S h T_\infty N_j dS$$

this can be expressed in matrix form as

$$([K] + [H])\{a\} = f$$

3.10 Time-Dependant (Transient) Problems

Many real problems are time-dependant. The finite element formulation of a transient problem is a relatively simple extension of the steady-state formulation. In addition to specified boundary conditions, transient problems require an initial condition (at time $t=0$) to be specified.

The transient heat conduction problem is given by equation 3.1b

$$k\nabla^2 T + Q = \rho c \frac{\partial T}{\partial t}$$

The procedure is similar to the steady-state problem. Using a weighted residual method and integrating by parts gives

$$\int_S q w dS + \int_A \left[k \frac{\partial w}{\partial x} \frac{\partial T}{\partial x} + k \frac{\partial w}{\partial y} \frac{\partial T}{\partial y} - Q w \right] dA + \int_A w \left(\rho c \frac{\partial T}{\partial t} \right) dA = 0 \quad (3.46)$$

The weighting function w is not a function of time. A finite element formulation is substituted for T , assuming that the time dependence can be separated from the spatial dependence

$$T = \sum_{i=1}^n N_i(x) T_i(t) \quad (3.47)$$

The weighting functions are equated to the shape functions (Galerkin's Method). Equation 3.47 is substituted into equation 3.46, and manipulated as before

$$\sum_{i=1}^n \int_A \rho c N_i N_j dA \frac{dT_i}{dt} + \sum_{i=1}^n \left[\int_A k \left(\frac{\partial N_i}{\partial x} \frac{\partial N_j}{\partial x} + \frac{\partial N_i}{\partial y} \frac{\partial N_j}{\partial y} \right) dA \right] T_i = \int_A Q N_j dA - \int_S q N_j dS$$

This can be written in matrix form as

$$[M] \{\dot{T}\} + [K] \{T\} = \{Q\} + \{q\} \quad (3.48)$$

$$\text{where } M_{ij} = \int_A \rho c N_i N_j dx dy$$

and the other terms are unchanged from the steady state formulation.

It is not possible to integrate the equations with respect to time analytically. So they are discretised in time by normal finite element techniques, i.e.

$$\{T\} = \sum N_n(t) T_n$$

If the temperature at the node changes from T_n to T_{n+1} over a time interval Δt , then the shape functions are given by (see Figure 3.1)

$$N_n = 1 - \xi$$

$$N_{n+1} = \xi$$

where ξ varies from 0 to 1 and is given by

$$\xi = \frac{t}{\Delta t}$$

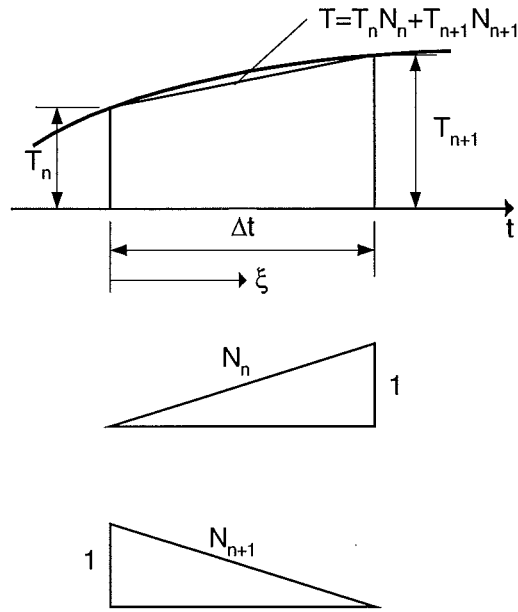


Figure 3.1. Temporal shape functions.

The derivatives of the shape functions with respect to time are

$$\dot{N}_n = \frac{-1}{\Delta t}$$

$$\dot{N}_{n+1} = \frac{1}{\Delta t}$$

Discretisation of equation 3.48 gives

$$[M] \left\{ T_k \dot{N}_k + T_{k+1} \dot{N}_{k+1} \right\} + [K] \{ T_k N_k + T_{k+1} N_k \} = \{ f \} \quad (3.49)$$

Substituting the expressions for the shape functions and their derivatives into equation 3.49 gives

$$\left(\frac{[M]}{\Delta t} + [K]\theta \right) \{T\}_{k+1} = \left(\frac{[M]}{\Delta t} - [K](1-\theta) \right) \{T\}_k + \{f\} \quad (3.50)$$

where

$$\theta = \frac{\int_0^1 W_j \xi d\xi}{\int_0^1 W_j d\xi}$$

Equation 3.50 is solved at each time step for the nodal temperatures at time $t_k = (k+1)\Delta t$.

Selection of different distributions for the weighting function, W_j , result in different values for θ , as shown in Figure 3.2.

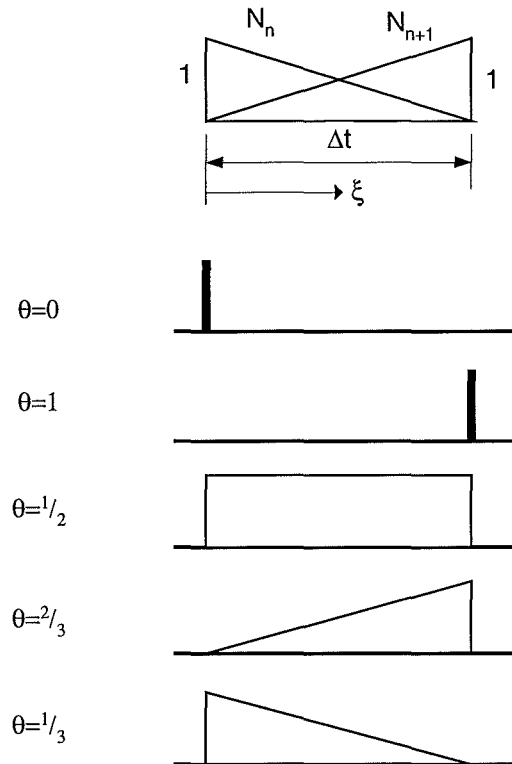


Figure 3.2. Weighting functions (Zienkewicz 1989).

It can be shown (Lewis et al 1996) that any scheme with $\theta \geq 1/2$ is unconditionally stable. If $\theta < 1/2$, then the scheme is stable subject to the condition

$$\Delta t \leq \frac{2}{(1-2\theta)\lambda_{\max}}$$

where λ_{\max} is the maximum eigenvalue of equation 3.48.

Although it is not unconditionally stable, the advantage of the forward difference scheme ($\theta=0$) is that it is explicit and computationally simple. Other schemes are implicit and require the solution of an equation system at each time step.

TASEF uses $\theta=0$ (Stern and Wickstrom, 1990). SAFIR allows the user to select the value of θ to be used (the parameter TETA in the DAT input file), and recommends a value of 0.9 (Nwosu et. al., 1999). Zienkiewicz (1989) reports that oscillations sometimes occur with $\theta=1/2$, and for a simple problem involving 10 quadratic elements to model transient heating of a bar it was found values of $2/3$ and 0.878 gave improved results.

3.11 Non-Linear Problems

Non-linear problems arise when k , ρ , and c are a function of the temperature. The matrix formulation is identical to equation 3.48, except an iterative solution is required at each time step. The simplest iterative method involves starting with an initial guess, T_0 , obtaining a more accurate solution by solving the equation

$$[M]\{\dot{T}\}_1 + [K(\{T\}_0)]\{T\}_1 = \{Q\}_0 + \{q\}_0$$

and then repeating the iteration scheme

$$[M]\{\dot{T}\}_n + [K(\{T\}_{n-1})]\{T\}_n = \{Q\}_{n-1} + \{q\}_{n-1}$$

until convergence lies within a suitable tolerance.

CHAPTER 4

SAFIR

4.1 Introduction

This chapter describes some aspects of finite element modelling with SAFIR that are not adequately covered in the SAFIR User Manual. Recommendations on mesh generation and node renumbering are given. The materials and time temperature curves available in SAFIR are described. This chapter should be useful to those using SAFIR by discussing material that is not covered elsewhere.

4.2 Modelling Considerations

4.2.1 Finite Element Mesh

In SAFIR, and in other finite element programs, a non-linear temperature field is approximated with a series of linear finite elements, so in general increasing the number of elements will decrease the overall error. However, increasing the number of nodes will increase the size of the conduction matrix, which will increase both the required storage and the computational effort required to solve the equations. For efficiency, it is recommended that larger elements are used where the temperature gradients are low, and smaller elements are used where the temperature gradient is high or non-linear. If it is not apparent by inspection where in the domain high temperature gradients can be expected, then it is generally better to analyse the model initially with a coarse mesh. Inspection of the results of the initial analysis will reveal the temperature gradients, which will indicate where the accuracy of the solution can be most efficiently improved by refining the mesh. Triangular or quadrilateral elements can be used to make a transition from a coarser to a finer mesh, typical mesh transitions are shown in Figure 4.1.

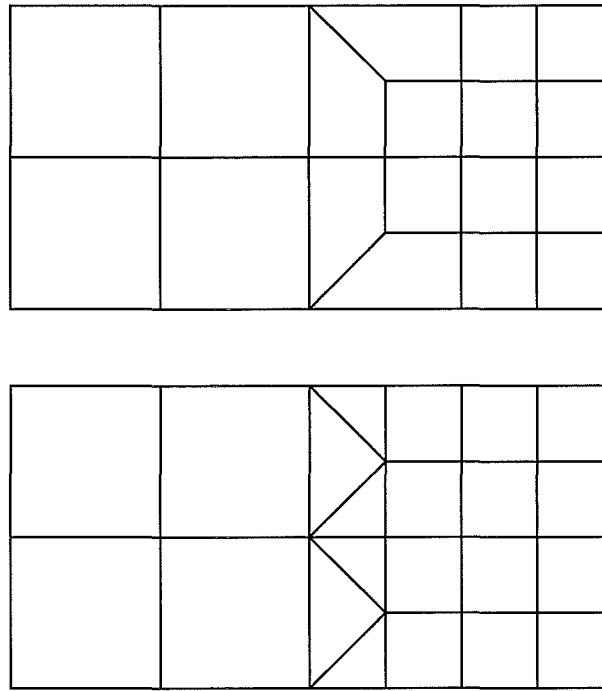


Figure 4.1. Use of quadrilateral and triangular elements in a transition from a coarse to a fine mesh.

General recommendations for the shape of finite elements are (Reddy & Gartling, 1994):

1. Aspect ratios should not be too high (3 is a reasonable upper limit).
2. Corners of quadrilateral elements should be kept as close as possible to 90° .
3. Internal angles in triangular elements should not be too acute.
4. The size of adjacent elements should not differ significantly.

These recommendations are made because structural elements tend to stiffen and lose accuracy if these recommendations are not followed. In thermal analysis these criteria are less critical, and the main criterion is to match the temperature gradient and the element size. So elements with large aspect ratios could be used without significant loss of accuracy if the temperature gradient is uni-directional. Of course, departing from these recommendations may not affect the accuracy of the solution if the area of interest is in a different part of the domain.

If a thermal and structural analysis is to be carried out on the model, then both temperature and stress gradients must be considered when the mesh is developed. For

example the temperature gradient will be highest near the soffit of a concrete slab subjected to a fire below it, but the stress gradient will be higher in the compressive region near the top of the slab.

4.2.2 Bandwidth and the Effect of Renumbering

In a large model typically many coefficients in the conductance matrix are zero. These values need not be stored and if this is considered in the computer model then the required space to store the matrix could be reduced, and the solution time could also be reduced.

Consider the four one-dimensional element model and associated “stiffness” matrix in Figure 4.1. Letters represent non-zero elements.

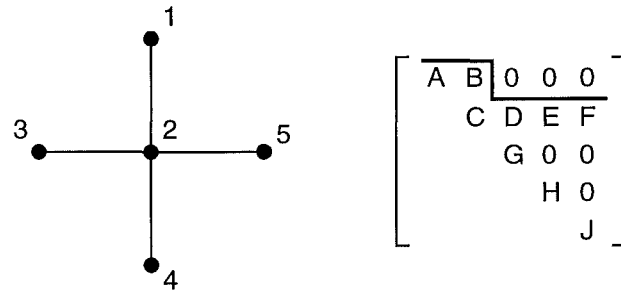


Figure 4.1. 5 node model and corresponding "stiffness" matrix (Cook et al 1989).

The line in the matrix that separates the zero entries at the top of the column from the non-zero entries is called the *skyline* or *profile*. The principal of the skyline technique is that only the portion of each column that is below the skyline is stored; this technique is used in SAFIR. The position of the skyline depends on the way the nodes are numbered, and it can be changed by changing the node numbering. If the skyline can be lowered then the storage requirements and solution time will be reduced. Consider the same finite element model as Figure 4.1 but with a different numbering sequence for the nodes (Figure 4.2)

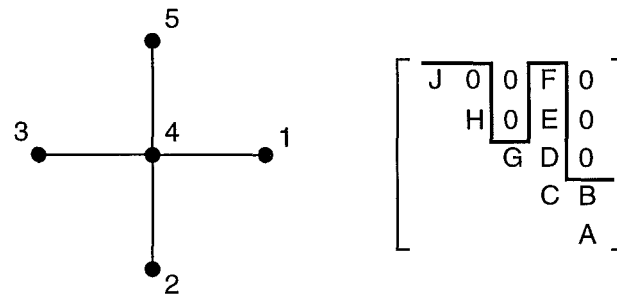


Figure 4.2. 5 node model and “stiffness matrix”, alternative numbering scheme (Cook et al 1989).

The numbering scheme in Figure 4.2 is preferred because the skyline is lower. This scheme is also better because zero entries below the skyline may be changed to non-zero by the equation solution process, and this matrix has no zero entries below the skyline.

In practice it is very difficult to predict the skyline when numbering a mesh manually. So most finite element programs include some sort of automatic renumbering scheme. The nodes are renumbered by the program before the matrices are solved, and then the mesh is restored to the original numbering so that the results can be easily interpreted. Some programs carry out the renumbering automatically, other programs give the user a choice of renumbering techniques. SAFIR lets the user choose between no renumbering, a geometrical renumbering scheme, a scheme called logical permutations, or both of the latter two.

4.2.3 Renumbering in SAFIR

SAFIR gives the user a choice of renumbering schemes. The scheme is specified in Series 9 of the DAT file.

No Renumbering

If the model does not have many nodes, then the computational effort required for renumbering may be greater than the computational effort saved in the solution process. In practice geometrical renumbering takes very little time, so it is recommended that this technique be used for all structures.

Geometrical Renumbering

Renumbering starts with a node which is specified by the user. Nodes are renumbered in sequence depending on the distance from the initial node. The choice of the initial node will affect the efficiency of this renumbering technique. SAFIR reports the length of the skyline in the .OUT file and on the screen, so the analysis may be interrupted after renumbering to check the effect of selecting different initial nodes.

Logical Permutations

In this technique, an image of the “stiffness” matrix is created. Instead of the actual values, the image contains either “false” if that entry is known to be zero or “true” if that entry is non-zero. SAFIR then changes the order of the equations to reduce the height of the skyline for the matrix image. When this scheme is chosen SAFIR stops after renumbering and stores the results of the renumbering in a text file. SAFIR must then be restarted and told to read the results of the previous renumbering from the file. This is done by specifying READRENUM in Series 9 of the DAT file.

4.3 Materials

4.3.1 Standard Properties

SAFIR has a number of material properties built into the program. Of the materials listed in the SAFIR manual, LWCONCSCH is not actually implemented in the program. There are two additional materials available that are not mentioned in the Manual. They are Type C Gypsum board and Type X Gypsum board. These were added to SAFIR for a special study (Milke & Dillon, 1997), but are valid for general use. Insulation is a user-defined material, but with properties that are independent of temperature. There is also one user-defined material with time dependent material properties. The materials available materials in SAFIR are summarised in Table 4.1.

Material Name	Description
STEELEC3	Steel, from Eurocode 3 (EC3, 1995)
STEELEC2	Cold worked reinforcing steel, from Eurocode 2 (EC2, 1993)
PSTEELA16	Cold worked prestressing steel, from Eurocode 2
CALCONCEC2	Concrete with calcareous aggregates, from Eurocode 2
SILCONCEC2	Concrete with siliceous aggregates, from Eurocode 2
CALCONCSCH	Concrete with calcareous aggregates, based on the Schneider model (Schneider 1988)
SILCONCSCH	Concrete with siliceous aggregates, based on the Schneider model
INSULATION	Material with constant thermal properties, specified by the user.
C GYPSUM	Type C Gypsum board
X GYPSUM	Type X Gypsum board
USER1	User-defined

Table 4.1. Materials available in SAFIR

4.3.2 Steel Materials

The same thermal properties are used for all steel materials. The density is 7850 kg/m^3 . Thermal properties are plotted in Figure 4.1.

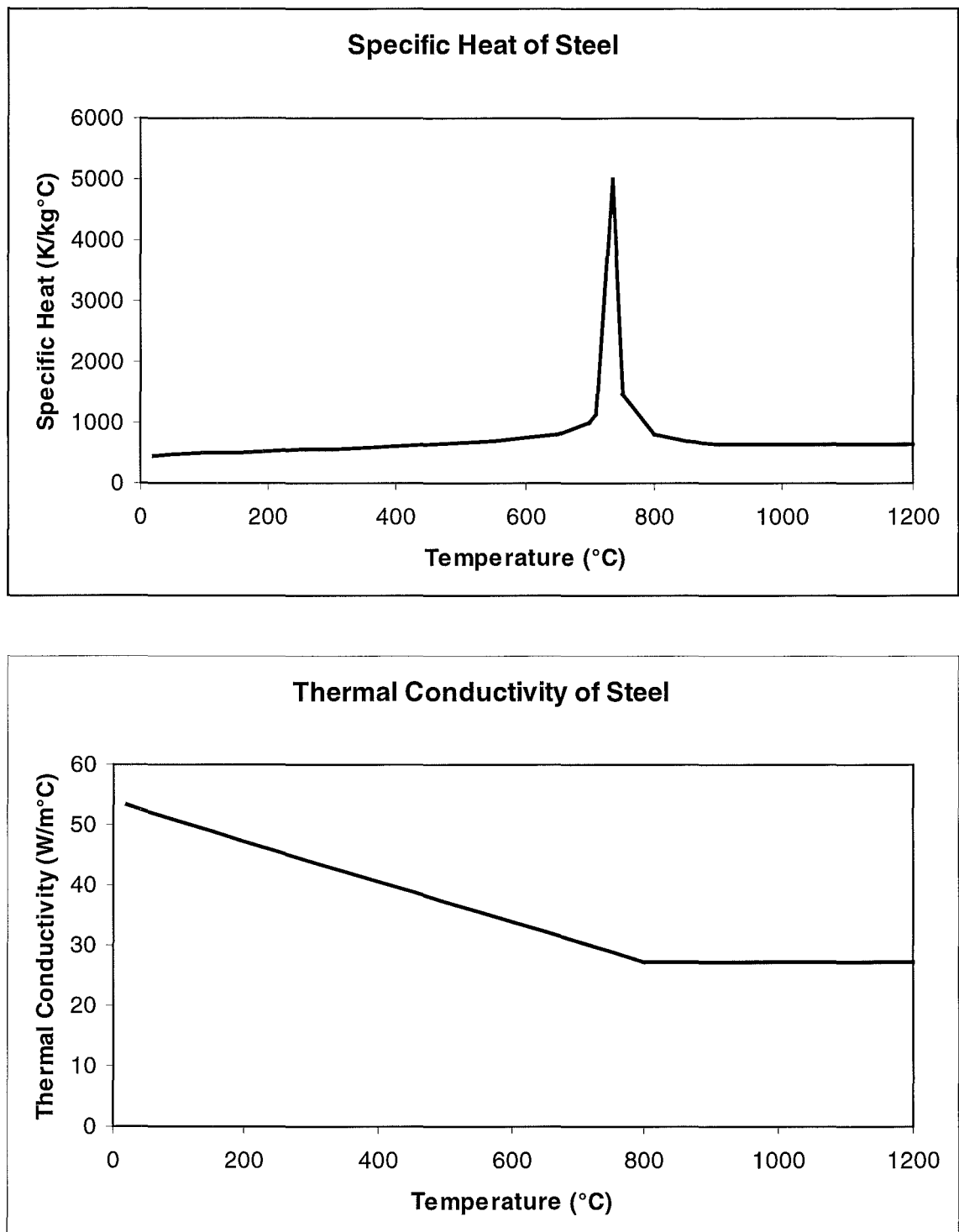


Figure 4.1. Thermal properties of steel in SAFIR.

4.3.3 Concrete Materials

There is no difference between the thermal properties of concrete calculated in accordance with Eurocode 2 and those calculated in accordance with Schneider (1988). The thermal conductivity of siliceous aggregate concrete and calcareous aggregate concrete are different, but the specific heat is the same for both concretes. The thermal conductivity used in SAFIR is independent of the moisture content, although the presence of water may actually increase the thermal conductivity slightly. The density of concrete is assumed to be 2208 kg/m^3 . The thermal properties used in SAFIR are shown in Figure 4.1 for a moisture content of 4% (the moisture content does not affect the thermal conductivity). If the concrete is in the cooling down phase, then the thermal conductivity is taken as the thermal conductivity corresponding to the maximum temperature experienced in the concrete. The specific heat in the cooling down phase depends on the maximum temperature reached in the concrete. If the maximum temperature has exceeded 200°C , then all the water is assumed to have evaporated and the curve is shown by the dashed line in Figure 4.1. If the maximum temperature has not exceeded 100°C , then the specific heat is given by the solid line. There is some interpolation made if the maximum temperature reached lies between these two values.

Note that the moisture content in SAFIR is specified in kg/m^3 . The moisture content for concrete in SAFIR is assumed to be 92 kg/m^3 unless specified otherwise in the data file.

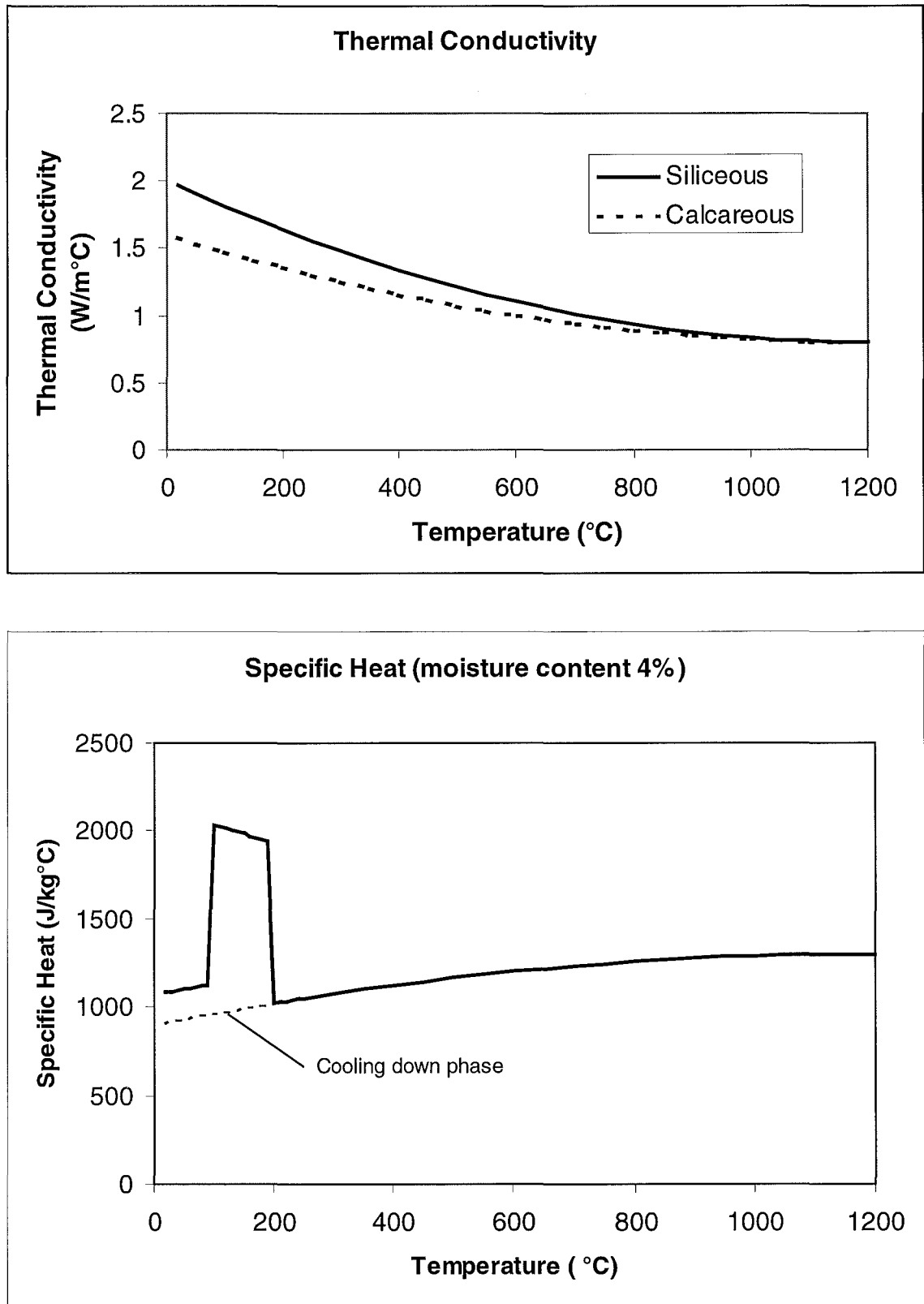


Figure 4.1. Thermal properties of concrete in SAFIR.

4.3.4 Gypsum

Type X Gypsum is a generic fire resistant gypsum board used in North America. It has some glass fibre reinforcing. Type C Gypsum is an improved formulation, with more glass fibre than Type X. Of the gypsum boards commonly used in New Zealand, Winstones GibBoard® (Winstones 1997) is a low density gypsum board with no reinforcing. Winstones Fyrelite® is a fibre reinforced gypsum board, similar in formulation to Type C Gypsum, although the comparison of the thermal properties has not yet been done. Thermal properties of Gypsum used in SAFIR are shown in Figure 4.1.

One of the criteria used for the performance of wall, floor, or ceiling systems is Integrity (SAA 1990). The barrier must not permit the passage of flame. Loss of integrity may occur by cracks opening in the board, gaps at the joints opening as the board shrinks, or the board falling off the framing when the fixings pull through the board. Gypsum lined walls with insulated cavities may lose their integrity sooner than walls with uninsulated cavities because the board on the side of the fire will heat up faster. Similarly, if gypsum board is used as insulation on a steel beam, deflection of the beam may compromise the integrity of the lining. It is difficult to model the integrity criteria, although some success has been made by adjusting the thermal properties to match test results (Thomas 1997). For this reason, caution should be used when modelling gypsum board beyond the range of known test performance. Thomas (1997) compares the thermal properties of gypsum wallboards reported by various researchers, and describes the physical changes undergone by gypsum as it is heated. The same reference also contains a useful summary of the thermal properties of timber for thermal modelling.

The moisture content of Type X Gypsum and Type C Gypsum in SAFIR is 17.5%. C Gypsum and X Gypsum have the same specific heat and thermal conductivity, but C Gypsum has a lower density.

The research for which the thermal properties of gypsum were added to SAFIR did not have to consider a cooling-down phase, so the properties are unchanged if the temperature is decreasing. This may lead to inaccurate results during the cooling down phase if the temperature has exceeded 100°C.

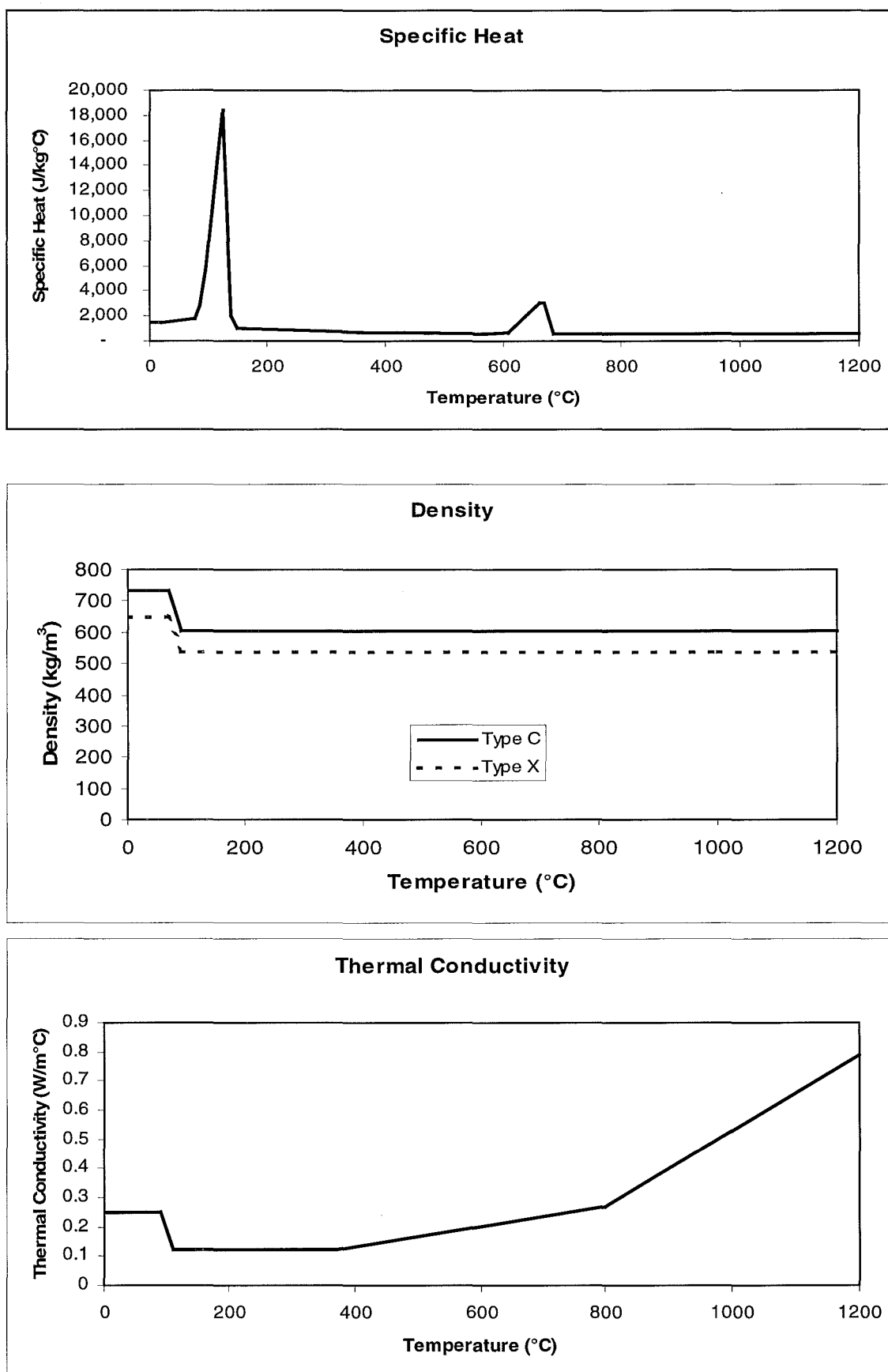


Figure 4.1. Thermal properties of Gypsum in SAFIR.

4.3.5 Accounting for Moisture Content in User-defined Materials

The moisture in gypsum board contributes significantly to its fire resistance, as a significant amount of energy is required to cause evaporation the water. Concrete also has a water content that affects its fire resistance, although to a lesser extent than in gypsum board. The presence of water will increase the thermal conductivity (although this is not taken into account in the Eurocode model for concrete), and the density will decrease as the water evaporates (again not included for concrete in SAFIR). The effect of the energy required to cause evaporation on the thermal analysis is normally accounted for by increasing the magnitude of the specific heat over the temperature range in which evaporation occurs. The graphs of specific heat for concrete (Figure 4.1) illustrate the effect. This same technique can be used to model moisture content in the user-defined material in SAFIR. The principle is illustrated in Figure 4.1.

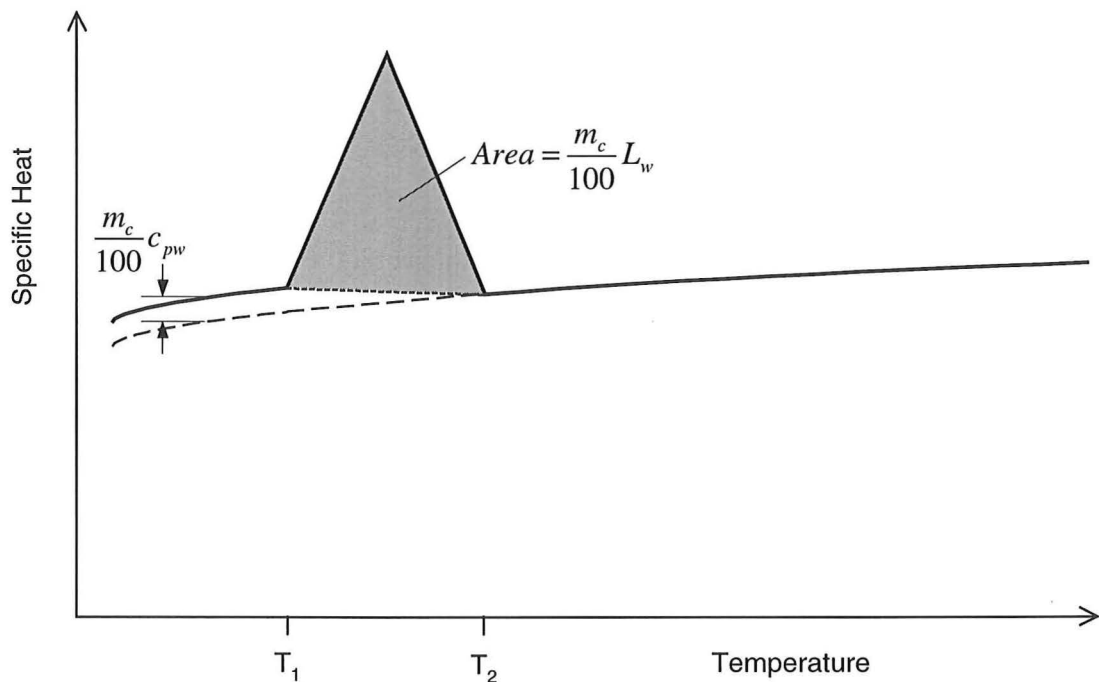


Figure 4.1. Modifying specific heat to account for moisture content. m_c is moisture content in %.

Below the temperature at which evaporation starts, the specific heat is increased by an amount which is proportional to the moisture content

$$\frac{m_c}{100} c_{pw}$$

where m_c is the moisture content of the material in percent, and c_{pw} is the specific heat of water (4180 J/kg°C). Over the temperature range where evaporation occurs, this increment is assumed to decrease linearly to a value of zero. The energy required to cause evaporation of the water is accounted for by increasing the specific heat over the temperature range where evaporation occurs.

In Figure 4.1 the shaded area is equal to

$$A = \frac{m_c}{100} L_w \quad (3.1)$$

So, if the profile of the curve over the evaporation temperature range is triangular, the peak value of the specific heat is

$$c_{p,peak} = \frac{(c_{p,T_1} + c_{p,T_2})}{(T_2 - T_1)} \frac{m_c}{100} L_w \quad (3.2)$$

where T_1 and T_2 are the lower and upper temperatures of the range over which evaporation occurs, c_{p,T_1} and c_{p,T_2} are the specific heats at temperatures T_1 and T_2 respectively, and L_w is the latent heat of evaporation of water (2260 kJ/kg).

There is no facility in SAFIR to define different thermal properties for the user-defined material during the cooling-down phase.

4.3.6 Specific Volume Enthalpy

Internally, SAFIR calculates the specific volume enthalpy defined as

$$e = \int_{T_0}^T \rho c_p dT$$

This ensures that a peak in the value of specific heat is not missed if consecutive time steps lie before and after the peak.

4.4 Time Temperature Curves in SAFIR

SAFIR has several standard time temperature curves that can be specified by the user. Two time temperature curves from test specifications are implemented, these are ISO 834 (ISO 1975), ASTM E119 (ASTM 1988). A hydrocarbon fire design curve from Eurocode 1 (EC1 1994) is also implemented. In SAFIR an ambient temperature of 20°C is assumed for the ISO 834 curve. The time temperature curves are shown in Figure 4.1. These time temperature curves are specified in SAFIR in Series 7 of the STR file with the entries ISO834, ASTME119, or HYDROCARB respectively.

As well as the standard temperature curves, a custom time-temperature curve may be specified by creating a comma delimited text file containing a series of time and temperature values, one per line. In these files the time should be in seconds, and the temperature in degrees celsius. These fires are specified by entering the name of the file in Series 7 of the STR file. The SAFIR Pre-Processor can create custom fire files for ISO 834 fires with linear decay, and for the Eurocode Parametric fire (EC1, 1994).

SAFIR will crash if element 1 does not have a time temperature curve specified for at least one of its faces. If the model cannot be renumbered so that one of the elements with a time temperature curve can be element number 1, then an F20 curve could be specified. This is a time temperature curve in SAFIR that specifies a constant temperature of 20°C, and has the effect of specifying that this element is exposed to an ambient temperature of 20°C.

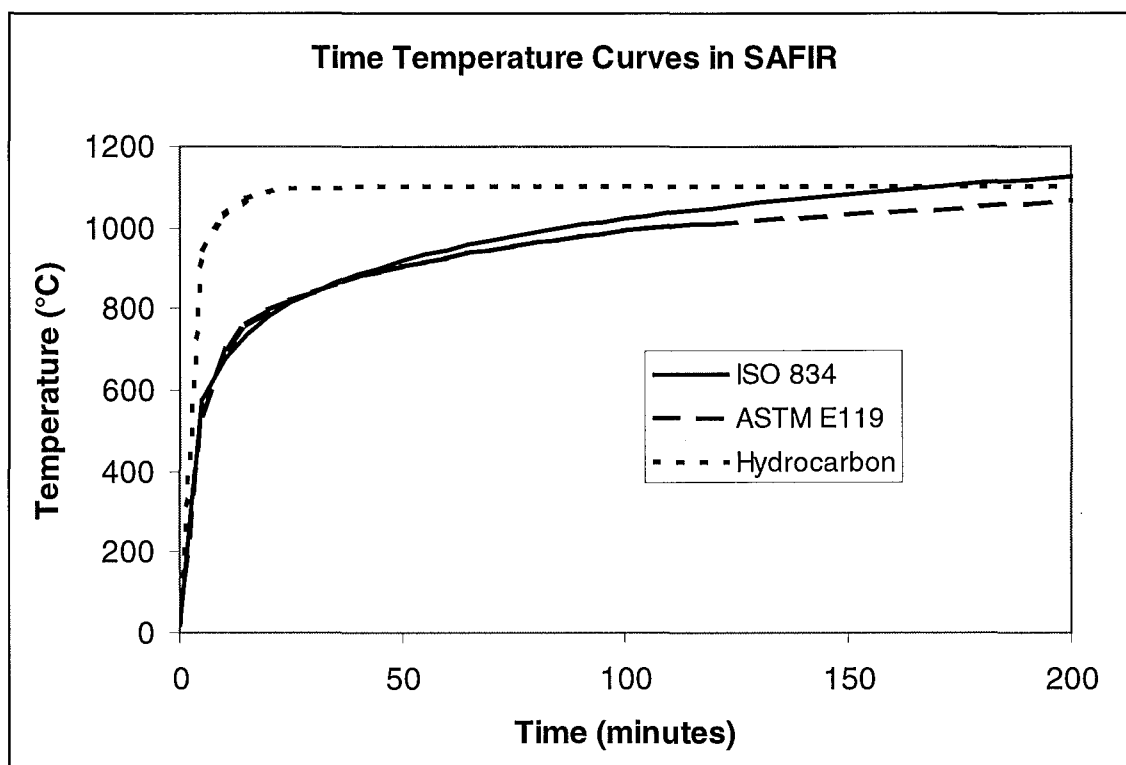


Figure 4.1. Time temperature curves in SAFIR.

4.5 Cavities

SAFIR can model the effect of internal cavities on the heat transfer process, modelling both convection and radiation, with the following restrictions. Cavities can only be included in two-dimensional models. Each cavity must be closed (completely surrounded by finite elements), and convex (no internal angle can exceed 180°).

The procedure used in SAFIR is the same as TASEF (Sterner and Wickstrom, 1990), and is based on Wickstrom (1979). It is summarised in the following sections.

4.5.1 Convective Heat Transfer in Cavities

The specific heat of air is typically three orders of magnitude lower than solid materials, so this is ignored in the model. It is also assumed that there is no flow of air into or out of the cavity. The temperature within the cavity is assumed to be uniform, and is determined by the total convective flux from the elements surrounding the cavity.

There is no heat gain or heat loss, so the total heat transferred must be zero:

$$\oint_s q_c dL = 0 \quad (3.3)$$

or, summing for each element surrounding the void:

$$\sum_{i=1}^n h_i (T_i - T_c) L_i = 0 \quad (3.4)$$

where h_i is the convective heat transfer coefficient associated with the element, T_i is the temperature of the face of the element adjacent to the cavity, T_c is the temperature within the cavity, and L_i is the length of the face of the element adjacent to the cavity.

The temperature of the face of each element is not uniform, and the temperatures are defined at the nodes. For linear elements the temperature of the face is taken as the average of the temperatures of the two nodes associated with that face. This relationship can be used to determine the temperature within the cavity.

4.5.2 Radiative Heat Transfer in Cavities

An energy balance for each surface of the cavity leads to the following equation:

$$\sum_{i=1}^N \left(\frac{\delta_{ij}}{\epsilon_i} - F_{j-i} \left(\frac{1 - \epsilon_i}{\epsilon_i} \right) \right) \frac{Q_i}{A_i} = \sum_{i=1}^N (F_{j-i} - \delta_{ij}) \sigma T_i^4 \quad (3.5)$$

where

j corresponds to each surface surrounding the void.

ϵ_i is the surface emissivity

A_i is the area of the surface, or for two-dimensional cavities, the length of the face

σ is the Stefan-Boltzmann constant

δ_{ij} is the Kronecker delta, defined as

$$\delta_{ij} = 1 \text{ when } i=j$$

$$\delta_{ij} = 0 \text{ when } i \neq j$$

$F_{i,j}$ is the view factor.

The view factor is defined as

$$F_{j-i} = \frac{1}{A_j} \iint_{A_j A_i} \frac{\cos \theta_j \cos \theta_i}{\pi S^2} dA_i dA_j \quad (3.6)$$

where S is the distance from a point on surface A_i to a point on surface A_j , and θ_i , θ_j are as shown in Figure 4.1.

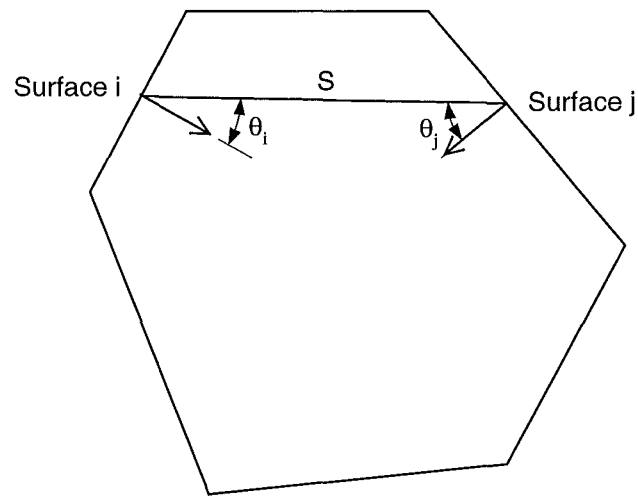


Figure 4.1. Notation for enclosure radiation (Wickstrom 1979).

For two-dimensional cavities with no shadowing surfaces (convex cavities) *Hottel's Crossed String Method* (Hottell 1954) is a simple way to calculate view factors (see Figure 4.2), and is used in both TASEF and SAFIR.

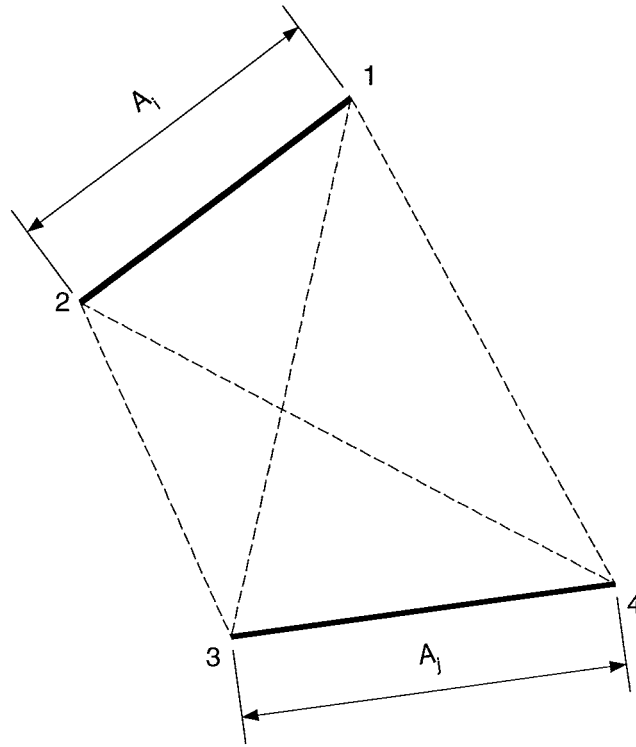


Figure 4.2. Hottel's Crossed-String Method for calculating view factors in a cavity.

$$F_{i-j} = \frac{L_{13} + L_{24} - L_{23} - L_{14}}{2A_i} \quad (3.7)$$

where L_{ij} is the distance between nodes i and j .

Equation 3.5 can be written in matrix form as

$$\{Q\}[X] = \sigma[Y]\{T\}^4 \quad (3.8)$$

where

$$X_{ij} = \left(\frac{\delta_{ij}}{\epsilon_j} - F_{i-j} \frac{1 - \epsilon_j}{\epsilon_j} \right)$$

$$Y_{ij} = (F_{i-j} - \delta_{ij})$$

Inverting $[X]$ in equation 3.8 lead to

$$\{Q\} = \sigma[X]^{-1}[Y]\{T\}^4 \quad (3.9)$$

The emissivity and cavity geometry are constant, so $[X]$ and $[Y]$ are constant, and the matrix need only be inverted once.

Equation 3.9 is used by SAFIR to calculate the heat fluxes to each surface in the cavity. When new nodal temperatures are calculated, either after the next iteration or next time step, a new set of fluxes is recalculated.

A soon to be released version of SAFIR has removed the restriction that voids must be closed and convex. This will make the modelling of cavity wall and ceiling systems easier.

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 Summary

The seemingly endless growth of the power of desktop computers is leading to the increasing use of finite element programs for engineering analysis. However, while the finite element method is a powerful and indispensable tool, the programs are too often used without proper understanding of the underlying theory and of the limitations of the software.

This report has reviewed the use of the finite element method for heat transfer analysis and shown how the variational formulation of the finite element method is derived from the classical heat transfer equations. It has discussed some of the features and limitations of SAFIR.

This report also presents a new Pre-Processor for SAFIR, which has a wider scope than the existing SAFIR Wizard.

5.2 Further Work

SAFIR is undergoing continued development, and the Pre- and Post-processors must be updated as necessary to reflect the capabilities of the new versions of SAFIR. The Pre-processor should be extended to handle reinforced concrete sections. It should be made capable of supporting three-dimensional thermal and structural models. It could also be improved by adding post-processing capability and the ability to run SAFIR from within the program, which would make modelling more convenient by providing all the necessary capability within in one program. Automatic mesh generation would further enhance the Pre-processor.

A comprehensive Users' Guide for SAFIR is needed to fully document the use of the program.

The removal of the restrictions on the void geometry in the next version of SAFIR will simplify the modelling of cavity wall systems and ceiling systems. It is

suggested that the limit on the number of user-defined materials in SAFIR be removed, or at least extended. This would make possible the modelling of timber structures, and other systems with materials that are not included in SAFIR.

REFERENCES

ASTM, 1988. *Standard Test Methods for Fire Tests of Building Construction and Materials*. E119-88. American Society for Testing and Materials.

Cook, R.D., Malkus, D.S., Plesha, M.E. 1989. *Concepts and Applications of Finite Element Analysis*. John Wiley & Sons.

EC1, 1994. *Eurocode 1: Basis of Design and Design Actions on Structures. Part 2-2: Actions on Structures Exposed to Fire*. ENV 1991-2-2. European Committee for Standardisation. Brussels.

EC2, 1993. *Eurocode 2. Design of Concrete Structures. ENV 1992-1-2: General Rules – Structural Fire Design*. European Committee for Standardisation. Brussels.

EC3, 1995. *Eurocode 3. Design of Steel Structures. ENV 1993-1-2: General Rules – Structural Fire Design*. European Committee for Standardisation. Brussels.

Franssen J.M. 1987. *Étude du Comportement du Feu des Structures Mixtes Acierbeton*. Thèse de Doctorat en Sciences Appliquées, NO. 111. Université de Liège, Belgium

Hottell, H.C. 1954 *Radiant Heat Transmission* in *Heat Transmission*, editor W. H. McAdams. McGraw-Hill Book Company. New York.

ISO, 1975. *Fire Resistance Tests – Elements of Building Construction. ISO 834 – 1975*. International Organization for Standardization, Berlin.

Lewis R.W., Morgan K., Thomas H.R., Seetharamu K.N. 1996. *The Finite Element Method in Heat Transfer Analysis*. John Wiley & Sons. England.

Milke J., Dillon S.E. 1997. *The Thermal and Structural Response of Light Steel Frame Wall Assemblies Exposed to Fire Using the SAFIR Finite Element Model*.

Nwosu D.I., Kodur V.K.R., Franssen J.M., Hum J.K. 1999. *User Manual for SAFIR A Computer Program for Analysis of Structures at Elevated Temperature Conditions*. National Research Council Canada. Internal Report 782.

Owen D.R.J. and Hinton E. 1980. *A Simple Guide to Finite Elements*. Pineridge Press. Swansea.

Reddy J.N. and Gartling, D.K. 1994. *The Finite Element Method in Heat Transfer and Fluid Dynamics*. CRC Press.

SAA, 1990. *Fire Resistance Tests of Elements of Structure*. AS 1530.4-1990. Standards Association of Australia.

Schleich J.B. 1987. *REFAO-CAFIR: Computer Assisted Analysis of the Fire Resistance of Steel and Composite Concrete-Steel Structures*. ECE Research 7210-SA/502, Final Report EUR 10828 EN, Luxembourg.

Schneider, U. 1988. *Concrete at High Temperatures – A General Review*. Fire Safety Journal, Vol. 13, No. 4.

Sterner E., Wickström U. 1990. *TASEF – Temperature Analysis of Structures Exposed to Fire – User's Manual*. Statens Provningsantalt. Sweden.

Thomas, G. C. 1997. *Fire Resistance of Light Timber Framed Walls and Floors*. Fire Engineering Research Report 97/7. University of Canterbury. New Zealand.

Wickström U. 1979. *A Numerical Procedure for Calculating Temperature in Hollow Structures Exposed to Fire*. Lund Institute of Technology, Sweden. Report No. 79-3.

Winstones. 1997. *Gib Fire Rated Systems*. Winstones Wallboards Ltd. New Zealand.

Zienkiewicz O.C. and Taylor R.L. 1989. *The Finite Element Method*. McGraw-Hill. London.

APPENDIX 1

SAFIR Pre-Processor Users' Manual

1 INTRODUCTION

SAFIR Pre-Processor is an interactive, graphical program for creating and editing SAFIR data files for two-dimensional heat transfer models.

Files can be created for models of steel sections with profile or box insulation, Dycore slabs, or Hi-Bond slabs. Any existing two-dimensional thermal model can be opened in the Pre-Processor and modified.

SAFIR Pre-processor is designed for thermal models only, so parameters in the data files not required for thermal analysis may not be correctly defined.

This manual assumes familiarity with Microsoft Windows and common Windows operation and nomenclature. The user should also be familiar with the SAFIR User Manual (Nwosu et. al., 1999).

A reference of the program functions is given in Section 2. Tutorials describing the steps in creating 3 different models are given in Sections 3 to 5.

1.1 Installation

The Pre-processor consists of the main executable file, and several files containing the definition of the steel sections, and Dycore slabs. All these files should be copied into the same directory, preferably the same directory as SAFIR.

1.2 DAT and STR filenames

Input data for a SAFIR model is defined in two files, the DAT file and the STR file. The name of the STR file is defined in the DAT file, which allows more than one DAT file to call the same STR file. This is an unnecessary complication. It is

recommended that the STR file is given the same name as the DAT file. The Pre-processor always does this, so the user is not given the opportunity to specify the name of the STR file. The program preserves the STR filename when opening existing files.

2 REFERENCE

2.1 Main Screen

The main program window is shown in Figure 1.

All program functions are available from the toolbar buttons, and are repeated in the menus. The toolbar buttons are grouped into similar functions.

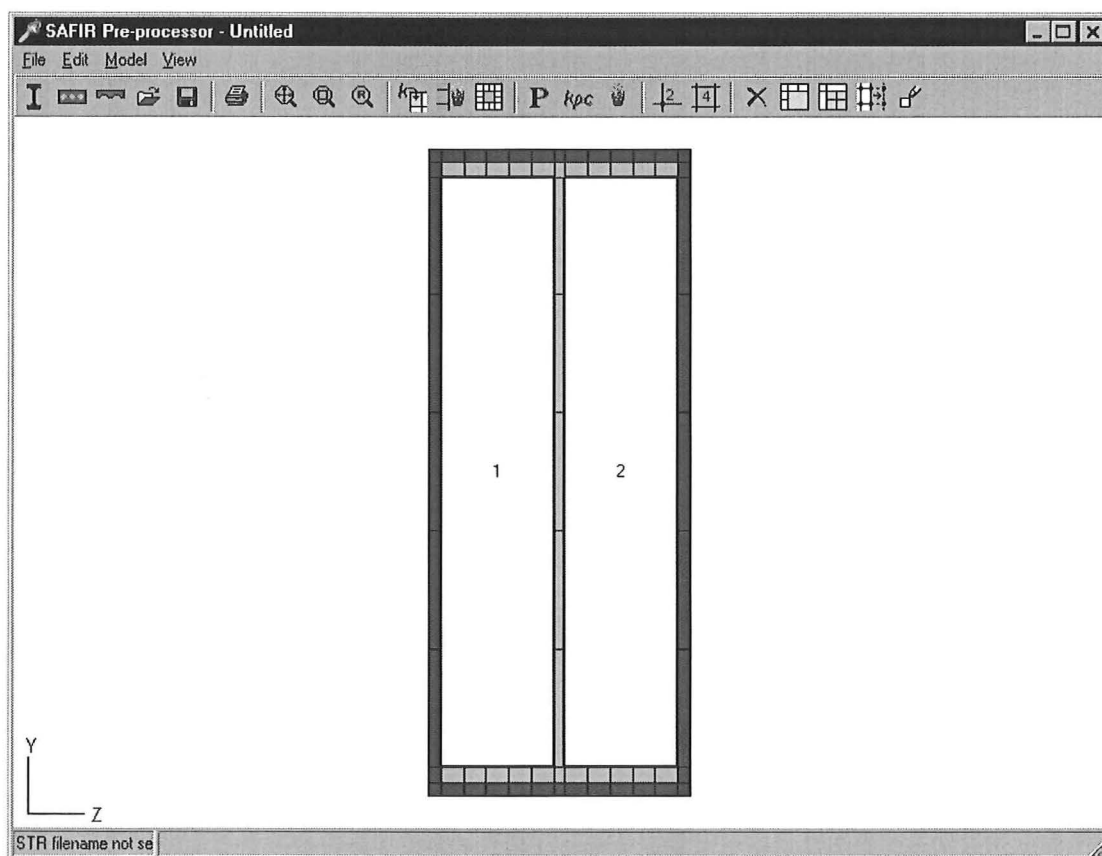


Figure 1. SAFIR Pre-Processor main window.

The main window shows the model. The axes are shown in the lower left-hand corner. Elements will be shown in different colours to identify their material. Faces to which

a fire has been applied will be outlined in red. The perimeter of internal cavities will be outlined in blue, and the void number will be shown within the void.

The functions of the buttons are summarised in Table 1.























Button	Function
	Create a new steel section model with or without insulation. Insulation may be to section profile or box-encasement.
	Create a new Dycore model
	Create a new Hi-Bond slab model
	Open an existing data file
	Save the current file
	Print the current model
	Zoom all
	Zoom window
	Redraw
	Assign defined materials to selected elements
	Assign defined fires to selected element faces
	Define a void
	Define SAFIR analysis parameters
	Define materials
	Define user fires
	Toggle display of node numbers
	Toggle display of element numbers
	Delete selected element(s)
	Merge selected elements into one element
	Split selected element(s) into multiple elements
	Copy the selected nodes to create new elements
	Draw a new element between existing nodes

Table 1. Button functions.

2.2 Coordinate System

Note that the SAFIR coordinate system differs from the normal convention. The first coordinate of a node lies on the vertical axis, and the second coordinate lies on the horizontal axis.

2.3 Steel Sections

Libraries of common steel section dimensions are built into the program. Sections available include common European and American I sections and also UB and UC sections available in New Zealand and Australia. The dimensions of the sections are defined in a series of text files (bhp.dat, hegeom.dat, ipegeom.dat, and american.dat), which files can be edited with a standard text editor. New sections can be added or dimensions of existing sections modified.

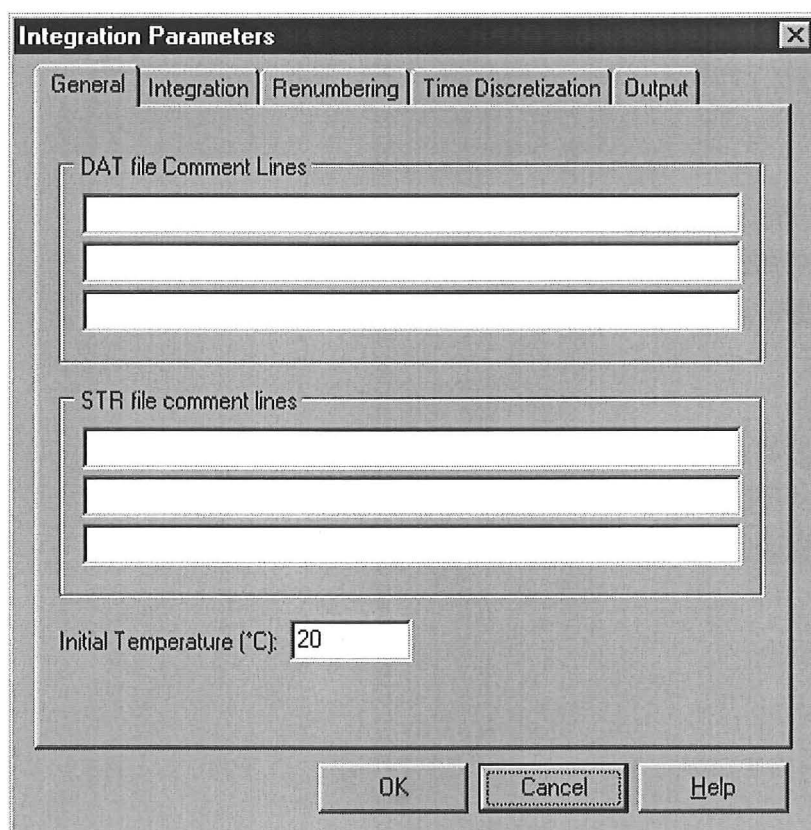
2.4 Selecting Elements

Some operations require the selection of elements. Elements can be selected by left-clicking once on the element, or by left-clicking and dragging a rectangle around the element on the screen. All elements completely within the rectangle when the mouse button is released will be selected. Elements that have been selected will be coloured yellow. Element selection is persistent until the operation is confirmed, so you can add elements to the group by selecting more elements. Selecting an element that has already been selected will remove it from the group. To complete a function right-click anywhere within the main window and select Ok from the pop-up menu. To cancel a function, right-click anywhere within the main window and select Cancel from the pop-up menu.

Some functions (eg assigning a fire curve to a face) require you to select the edge of a group of elements by dragging a rectangle. Any edges completely within the rectangle will be selected. Edges that have been selected will show in yellow.

2.5 Defining Analysis Parameters

Click the Analysis Parameters button. In the dialog box select each one of the tabs in turn and enter the required parameters. In the following description of the parameters, the corresponding SAFIR parameter name used in the SAFIR User's Manual is shown in capital letters in brackets.



The image shows a software dialog box titled "Integration Parameters". It has five tabs: "General", "Integration", "Renumbering", "Time Discretization", and "Output". The "General" tab is currently selected. Inside the dialog, there are two groups of text input fields. The first group is labeled "DAT file Comment Lines" and contains three empty text boxes stacked vertically. The second group is labeled "STR file comment lines" and also contains three empty text boxes stacked vertically. Below these groups is a single text input field labeled "Initial Temperature (°C):" with the number "20" entered. At the bottom of the dialog are three buttons: "OK", "Cancel", and "Help".

Figure 2. Integration parameters dialog box.

General Tab

Enter up to three lines of text for each of the data files to describe the model. These lines will be inserted at the top of the DAT and STR files. Enter the initial temperature (TINITIAL), the default value is 20°C.

Integration Tab

Enter the following parameters:

Time integration parameter (TETA), defaults to 0.9.

Maximum length of K11 (ILARGEUR11), defaults to 50,000.

Maximum length of K12 (ILARGEUR12), defaults to 1,000.

Precision (PRECISION), defaults to 0.001.

Number of integration points for the elements (NGSOLID), defaults to 2.

Number of integration points for the structure (NPTOT), defaults to 2.

Renumbering Tab

Specify which renumbering scheme (if any) is to be used. The options are no renumbering (NORENUM), renumbering by logical permutations (RENUMPERM), geometrical renumbering (RENUMGEO), both geometrical renumbering and renumbering by logical permutations (RENUM), or specify that SAFIR is to read the renumbering scheme from a previous logical permutation renumbering (READRENUM).

If you select to Logical Permutations, then the SAFIR analysis stops after renumbering. You must then modify the DAT file to specify READRENUM in Series 9 of the DAT file, and run the analysis again.

Time Discretization Tab

Specify a series of time steps and the time limit up to which that time step will apply (at least one pair of data). This corresponds to the TIME... ENDTIME section of the DAT file.

Output Tab

Select one of the following:

- Store average temperatures of elements (MAKE.TEM),
- Store temperatures of nodes (MAKE.TND), or
- Store temperatures of the first NNODE/2 nodes (MAKE.TSH).

The second option should be chosen if you want to plot temperature contours with DIAMOND97. The other options apply if a subsequent structural analysis is to be carried out.

Results: check one or more of the following:

- Write temperature variation at every time iteration (PRINTDEPL)
- Write out of balance forces at every iteration (PRINTDHE)
- Write reactions (PRINTREACT)

It is not usually necessary to select any of these. The last two are not applicable to pure thermal analyses.

Store Temperature in File (CFILENAME): specify the filename where the temperatures are to be written. It is not necessary to specify a file extension, TND, TSH, or TEM will be added as an extension as appropriate.

Timestep for results (TIMEPRINT): specify the time interval at which the temperatures will be written.

2.6 Defining Materials

Click the Define Materials Properties button.

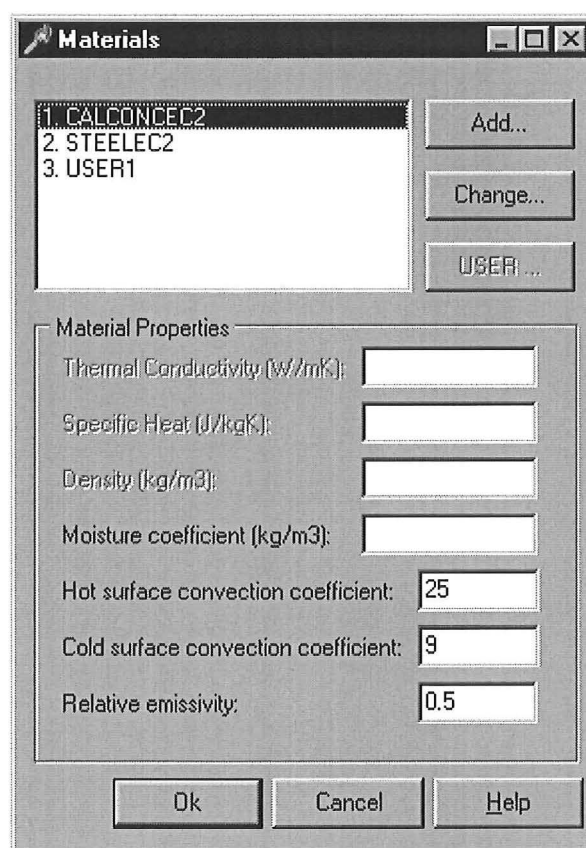


Figure 3. Defining material properties.

The materials that are already available in the model will be listed.

Click Add.. to add a material. Click Change.. to change a material. Note that only one user defined material (USER1) should be added. All materials recognised by SAFIR are available for use, but the thermal properties of all the steel materials are identical. Similarly the thermal properties of both siliceous aggregate concretes are the same, and the thermal properties of both calcareous aggregate concretes are the same.

For each material, add or edit the thermal properties in the text boxes. Properties that are not appropriate for a particular material cannot be entered.

For the User-defined material, enter the convection coefficients and emissivity in the text boxes. Click USER... to display a dialog box where you can enter the specific heat (in J/kg K), thermal conductivity (in W/m K), and density (in kg/m³) as a function of temperature. SAFIR will linearly interpolate between the values you enter. To display a chart of any of these properties, select one of the radio buttons above the chart.

2.7 Assigning Materials to Elements

To assign any material to an element or elements, click the Assign Materials to Elements button, select the elements, right-click and select Ok from the pop-up menu. To complete the assignment, select the desired material from the dialog box. The dialog box will show the materials that you have already defined. To add or modify materials, click the Materials... button, and the Define Materials dialog box (see Section 2.6) will be displayed.

2.8 Assigning Standard Fire curves

To assign one of the standard SAFIR fire curves (ISO 834 or ASTM E119) to an element face, click the Assign Fire button, select the element edges, right-click and select Ok from the pop-up menu, select the fire curve and click Ok. You can also assign a user-defined fire if you have previously created one. You can remove a boundary condition from an element face by selecting None.

2.9 User-defined fires

SAFIR lets you apply a user-defined fire curve. SAFIR Pre-processor can generate user-defined fire files for an ISO 834 fire with a linear decay, and a Eurocode parametric fire.

To define a user fire, click the Define User Fire button.

For the ISO834 fire with linear decay, you must specify the time to the maximum temperature (in minutes) and the decay rate (in °C/hour). For a Eurocode fire you must specify the compartment opening factor ($m^{1/2}$), thermal conductivity (W/mK), density (kg/m^3), specific heat (J/kgK), and fuel load (MJ/m^2).

User Defined Fire	
Fire Curve <input checked="" type="radio"/> ISO with linear decay <input type="radio"/> Eurocode Parametric Fire	
Time to maximum temp.: 75 mins. Maximum temperature: 978 °C Total duration: 266 mins. Decay rate: 300 °C/hour	
Parameters Time to maximum temp (min): 75 Decay rate (°C/hour): 300 Opening Factor ($m^{1/2}$): Thermal conductivity (W/mK): Density (kg/m^3): Specific Heat (J/kg K): Fuel Load (MJ/m^2 total area):	
OK Cancel Help	

Figure 4. Parameters for user-defined fire.

The time to maximum temperature, maximum temperature, total fire duration, and decay rate are calculated as you enter the parameters and shown in the lower left of the dialog box. The figures will be updated by clicking in any text box other than the one you are editing.

The Eurocode Parametric fire has a decay rate of 625 °C per hour, modified for opening factor and thermal insulation by (recommended by Buchanan 1999)

$$\frac{dT}{dt} = 625 \left(\frac{F_v}{0.04} \right) \frac{\sqrt{k\rho c}}{1160}$$

The time and temperature data for a user-defined file are saved in a text file of the same name as the DAT file, but with a FIR extension. This is a comma-delimited file consisting of time and temperature data. You may define any fire curve by creating an appropriate file with a text editor. SAFIR will linearly interpolate between the temperatures for intermediate times.

2.10 Deleting elements

Click the Delete button, select the elements as described in Section 2.3, right-click and select Ok to confirm the deletion.

2.11 Merging elements

Click the merge elements button, select the elements by dragging a rectangle to completely enclose the elements to be merged (you cannot select individual elements with this function). The elements to be merged should form a rectangle. The elements will be merged as soon as you release the mouse button, there is no right-click needed.

2.12 Splitting elements

An element may be split into more than one element with this tool. Click the Split button, select the elements as described in Section 2.3, right click and select Ok to confirm the action. You are prompted for the number of divisions along each axis. For example to split the element in half along in both directions, enter 1 division for both axes. Click Ok to carry out the split.

Note that only rectangular elements can be split. The operation will not be successful on triangular and non-rectangular quadrilateral elements.

2.13 Drawing new elements

Click the Draw New Element button. Draw the element by clicking on the nodes in an anti-clockwise sequence. Complete the element by clicking again on the first node.

2.14 Defining voids

Click the Define Voids button. Specify the void number to be defined (select an existing void number to redefine that void). Define the void by clicking on the nodes at the corners of the void. Click on the first node to complete the void.

3 TUTORIAL 1

In this tutorial we will create a model of a 610UB101 steel beam, with box insulation of 12 mm of Type C Gypsum. An ISO 834 fire curve will be applied to the sides and base.

3.1 Define the section

Click the new steel section button. Select BHP in the library list box, and then select 610UB101 from the section list box. Click the Box radio button and enter a thickness of 12 mm. Click OK.

3.2 Define the fire curve

Click the Assign Temperature Curve button. Click and drag a rectangle around the exterior faces of the insulation elements on the sides and bottom of the model (see Figure 5).

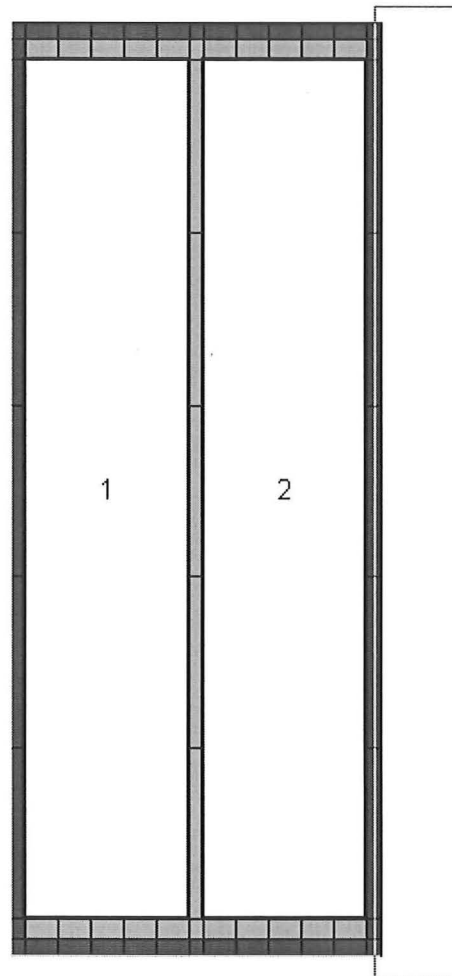


Figure 5. Selecting element edges to apply a temperature curve.

When finished, right-click and select OK to confirm. Select ISO and click OK to apply the temperature curve. The side and bottom edges of the model should be highlighted in red.

If you have selected too many edges, then click the Assign Temperature Curve button again, and drag a rectangle around those edges you want to remove the applied temperature from. Right-click, select None, and click OK. This should remove the temperature curve from those edges.

3.3 Analysis parameters

Click the Analysis Parameters button.

On the General tab, enter up to three lines of comment for the DAT and STR data files. Set the initial temperature to 20°C.

Select the Integration tab. The default values of these parameters need not be changed for this model.

Select the renumbering tab. Select the *Geometrical Method* of renumbering, and enter 1 as the starting node.

Select the Time Discretization tab. Enter the following values of Time Step and Upperlimit:

Time Step	Upperlimit
10	500
20	1000
50	2000

Select the Output tab. Select *Store Temperatures of Nodes*, make sure none of the three Results check boxes are checked. Enter a filename of tutorial1 for the nodal temperatures. Enter a timestep of 30.

Click OK.

3.4 Materials

The default material for the insulation is Insulation. We want to change this to Type C Gypsum.

Click the Define Material Properties button. Select Insulation and click the Change button. Select C Gypsum from the available materials and click OK. The defaults values for convection coefficients and emissivity are acceptable so click OK.

3.5 Save the file

Select File|Save As and save the file as Tutorial1.

4 TUTORIAL 2

In this tutorial, we will create a steel beam and concrete slab. The steel beam will be profile insulated, and will have a user defined fire applied to the exposed faces. We will also refine the mesh for the web elements.

4.1 Define the section

Click the new steel section button. Select a 610UB101 from the BHP library. Select Profile insulation with a thickness of 12 mm. Click OK. This should give you the section shown in Figure 6.

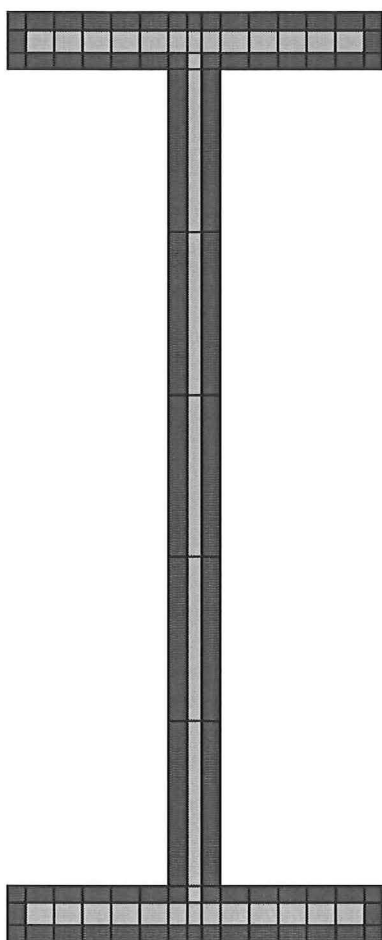


Figure 6. 610UB101 with profile insulation.

4.2 Refine the mesh in the web

Click the Split Elements button, and drag a rectangle that encloses only the elements in the web of the section (see Figure 7). Alternatively you can select the web elements by clicking on each element in turn.

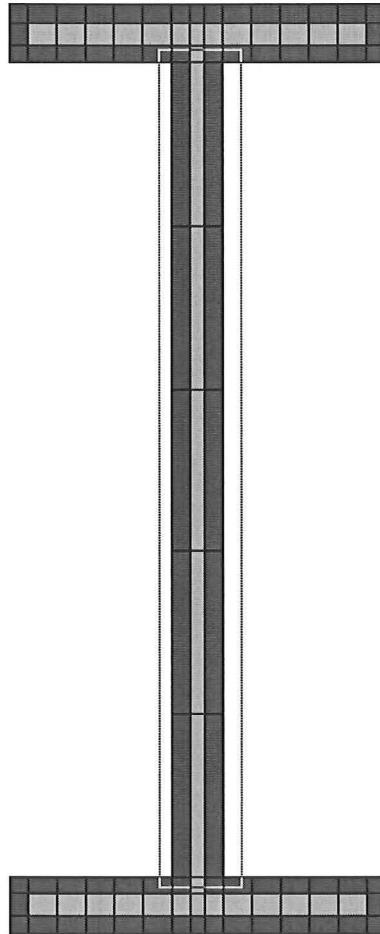


Figure 7. Selecting the web elements.

Right click and select OK. If you inadvertently select too many elements, you can deselect an element by clicking on it.

In the Split Elements dialog box, select 2 divisions along the Y axis, and 0 divisions along the Z axis. Click OK. You should now have a section that looks like Figure 8.

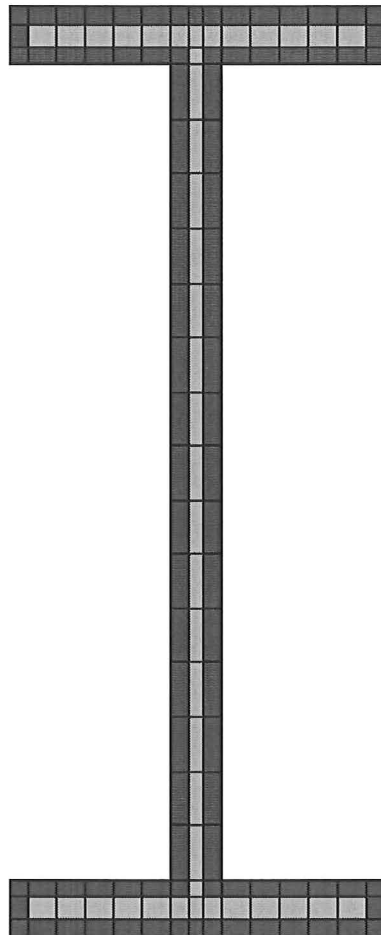


Figure 8. Model with refined mesh for web elements.

4.3 Delete the unnecessary insulation elements

We will now delete the unneeded elements along the top of the steel section.

Click the delete elements button. Drag a rectangle around the insulation elements above the top flange (or click on each element in turn). Right-click and select OK. You should now have a section that looks like Figure 9.

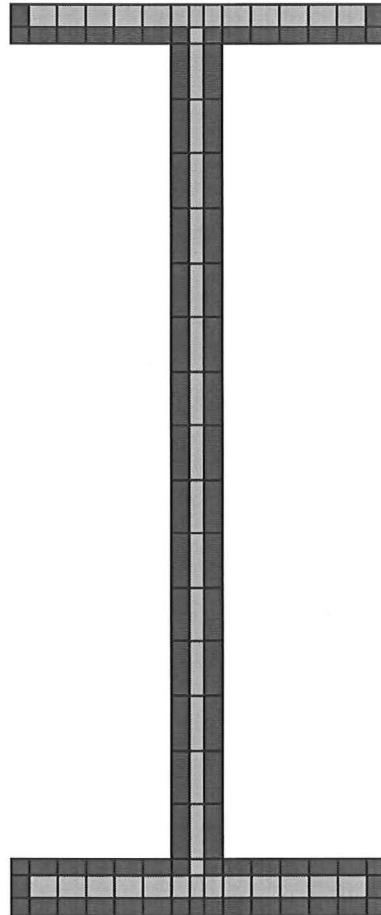


Figure 9. Unneeded insulation elements deleted.

4.4 Create the slab

Select the Extrude Element Edges button, and drag a rectangle around the top edges of the elements that lie at the top of the model. Right click and select OK. Enter a distance of 0.150 m and click OK. Some of the section will probably lie outside the

view window, so click the Zoom All button to display the whole section. The model should look like Figure 10.

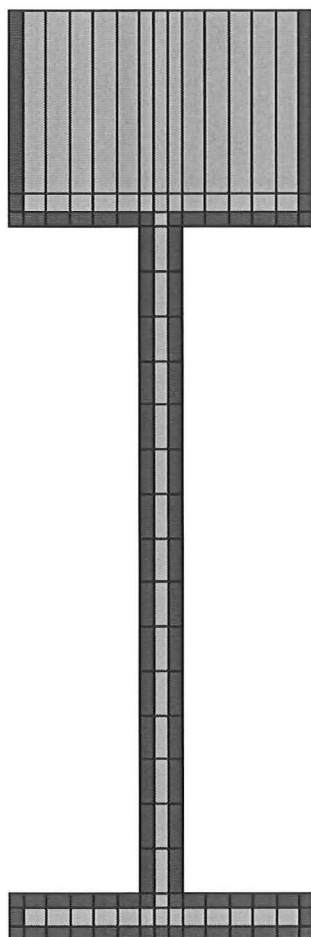


Figure 10. Top edge extruded to create slab.

To complete the slab, click the Extrude Element Edges button, and select the right and left edges of the slab. Do not include the insulation on the flange in your selection. Right-click and select OK. Enter a distance of 0.5 m and click OK. Your model should now look like Figure 11.

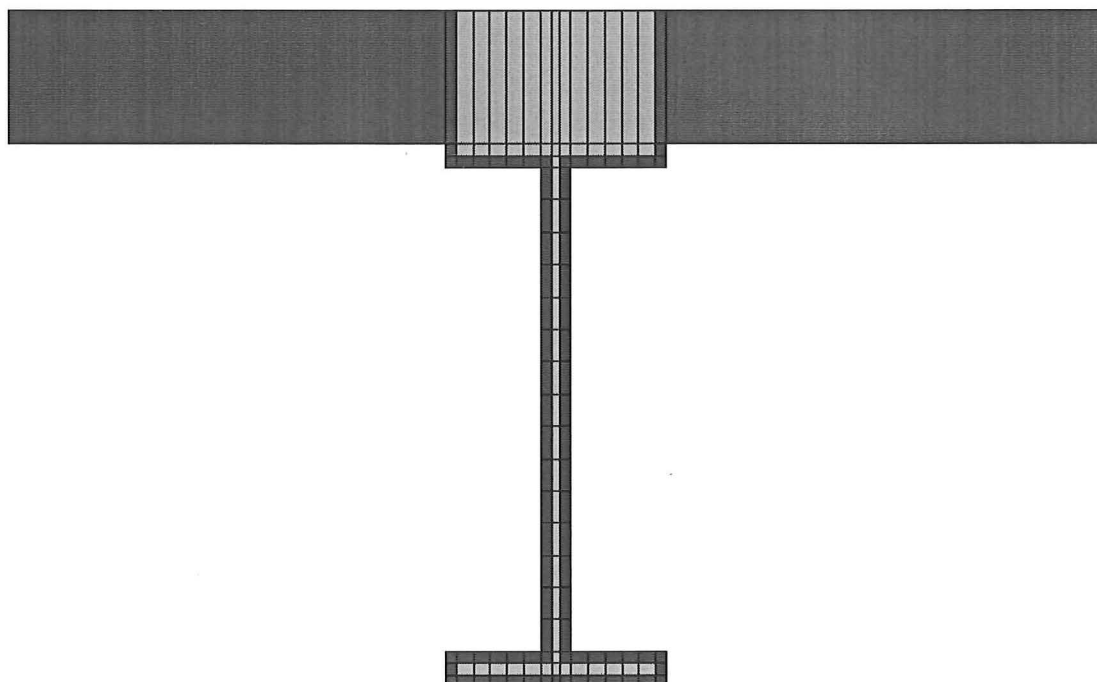


Figure 11. Slab geometry defined.

4.5 Define slab material

Click the Define Material Properties button. Click Add, and add CALCONCEC2. Enter a moisture content of 90 kg/m^3 and click OK.

Click the Assign Materials to Elements button. Select all the slab elements, right-click and select OK, and then assign the CALCONCEC2 material. The slab elements should now be plotted in a different colour.

4.6 Refine the slab mesh

Click the Split Elements button. Select the slab elements to the left and right of the steel section. Right-click and select OK. Select 0 divisions along the Y axis, and 10 divisions along the Z axis, and click OK. The model should now look like Figure 12.

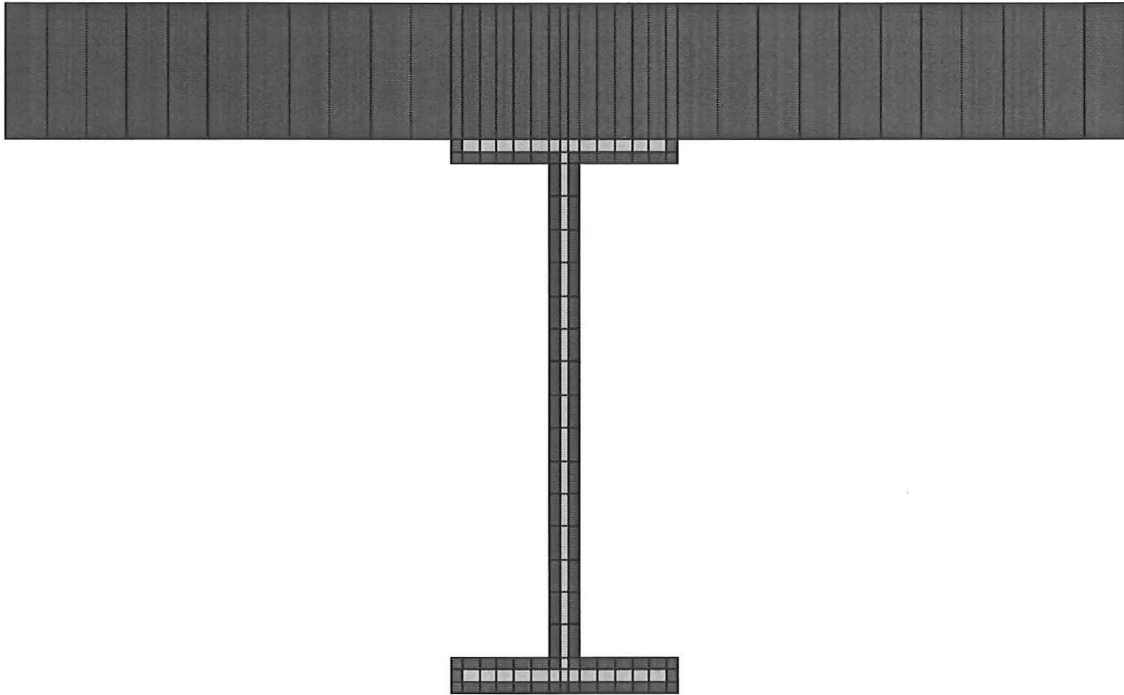


Figure 12. End of first step in slab mesh refinement.

Finally, split all the slab elements again, this time into 4 divisions on the Y axis, and 0 divisions on the Z axis. The model should now look like Figure 13.

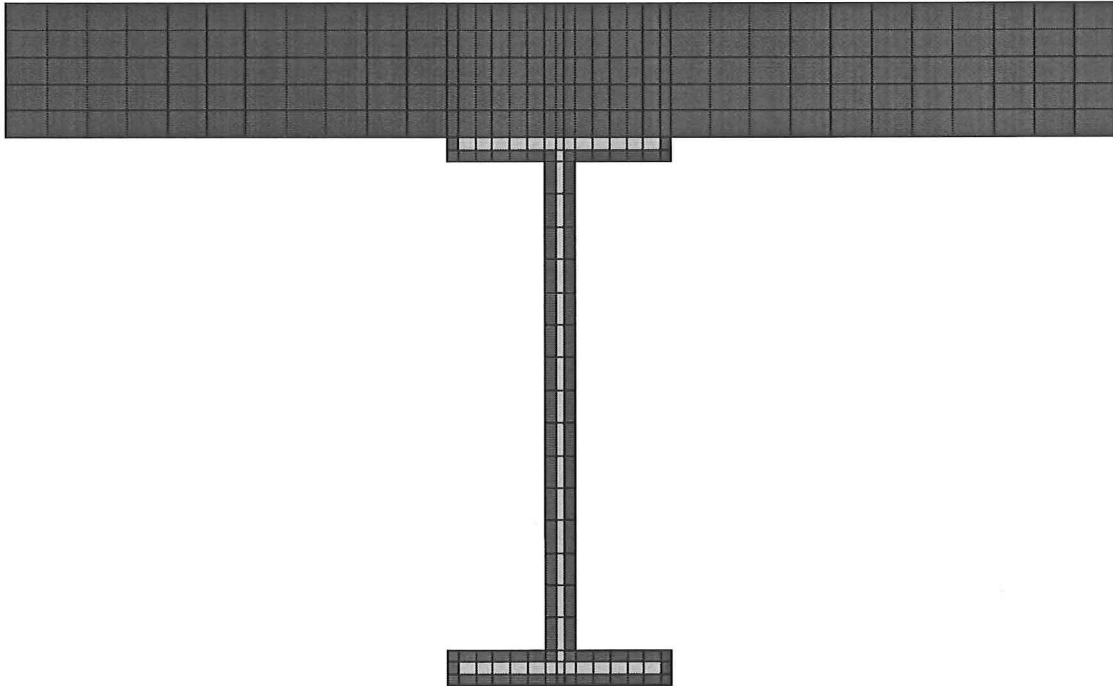


Figure 13. Slab mesh definition completed.

4.7 Complete the model

Complete the model by changing the default Insulation material to Type C Gypsum, apply an ISO 834 fire to the outer edges of the gypsum, and define the analysis parameters.

5 TUTORIAL 3

In this tutorial, we will create a model of a steel beam with concrete insulation between the flanges.

Click the new steel section button. Select a 610UB101 from the BHP library. Do not apply any insulation. Subdivide the web elements as in Tutorial 2, to give a section as shown in Figure 14.

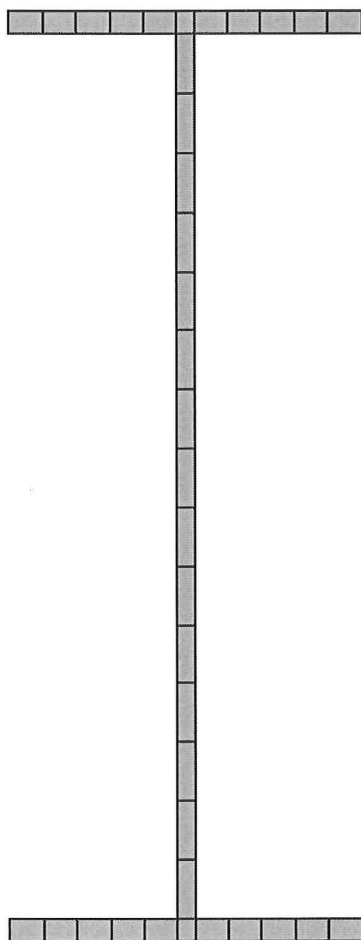


Figure 14. 610UB101.

To create the concrete infill, click the Draw New Element Button, and draw two new elements that completely fill the space between the flanges by clicking on the nodes to define the corners of the new elements in an anti-clockwise sequence. The model should now look like Figure 15.

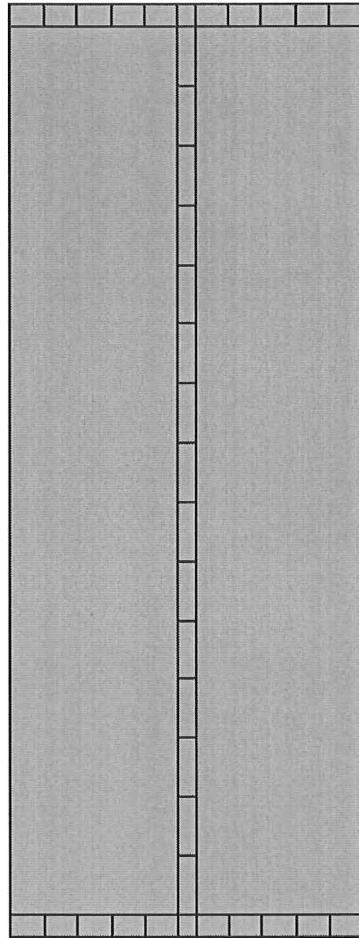


Figure 15. New element created between flanges.

Click the Define Material Properties button, and add CALCONCEC2 to the list of materials. Click the Assign Materials to Elements button, select the two new elements, and assign the CALCONCEC2 material to these elements.

Finally, click the Split Elements button, select the two concrete elements, and select 14 divisions on the Y axis, and 4 divisions on the Z axis. The completed model should look like Figure 16.

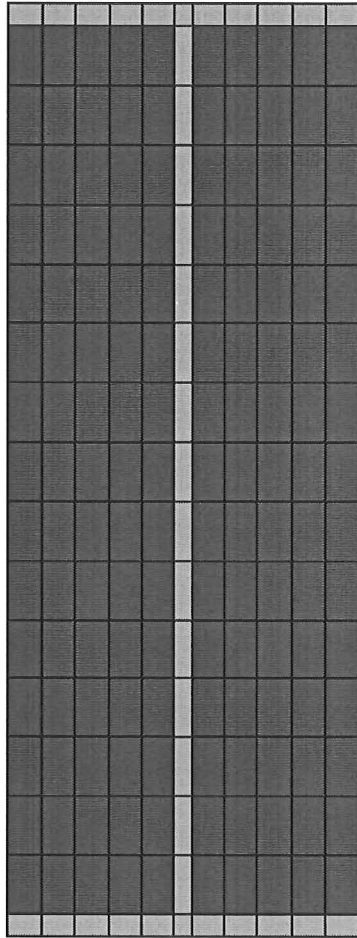


Figure 16. Completed model.

5.1 Define and Apply a User-defined fire

Click the User-Defined Fire button. Select an ISO Fire with Linear Decay. Enter 60 minutes for the Time to Maximum Temperature and a decay rate of 120°C/hour. Click OK.

Click the Assign Temperature Curve to Element Faces button. Drag a series of rectangles around the outer edges of the outer elements. Right-click and select OK. We have already defined a User-Fire, so one of the options will now show ISO Fire With Linear Decay. Check this fire, and click Ok to assign that fire to the model.

6 REFERENCES

Buchanan A.H. 1999. *Structural Design for Fire*. Unpublished.

Nwosu D.I., Kodur V.K.R., Franssen J.M., Hum J.K. 1999. *User Manual for SAFIR A Computer Program for Analysis of Structures at Elevated Temperature Conditions*. National Research Council Canada. Internal Report 782.

FIRE ENGINEERING RESEARCH REPORTS

95/1	Full Residential Scale Backdraft	I B Bolliger
95/2	A Study of Full Scale Room Fire Experiments	P A Enright
95/3	Design of Load-bearing Light Steel Frame Walls for Fire Resistance	J T Gerlich
95/4	Full Scale Limited Ventilation Fire Experiments	D J Millar
95/5	An Analysis of Domestic Sprinkler Systems for Use in New Zealand	F Rahmanian
96/1	The Influence of Non-Uniform Electric Fields on Combustion Processes	M A Belsham
96/2	Mixing in Fire Induced Doorway Flows	J M Clements
96/3	Fire Design of Single Storey Industrial Buildings	B W Cosgrove
96/4	Modelling Smoke Flow Using Computational Fluid Dynamics	T N Kardos
96/5	Under-Ventilated Compartment Fires - A Precursor to Smoke Explosions	A R Parkes
96/6	An Investigation of the Effects of Sprinklers on Compartment Fires	M W Radford
97/1	Sprinkler Trade Off Clauses in the Approved Documents	G J Barnes
97/2	Risk Ranking of Buildings for Life Safety	J W Boyes
97/3	Improving the Waking Effectiveness of Fire Alarms in Residential Areas	T Grace
97/4	Study of Evacuation Movement through Different Building Components	P Holmberg
97/5	Domestic Fire Hazard in New Zealand	KDJ Irwin
97/6	An Appraisal of Existing Room-Corner Fire Models	D C Robertson
97/7	Fire Resistance of Light Timber Framed Walls and Floors	G C Thomas
97/8	Uncertainty Analysis of Zone Fire Models	A M Walker
97/9	New Zealand Building Regulations Five Years Later	T M Pastore
98/1	The Impact of Post-Earthquake Fire on the Built Urban Environment	R Botting
98/2	Full Scale Testing of Fire Suppression Agents on Unshielded Fires	M J Dunn
98/3	Full Scale Testing of Fire Suppression Agents on Shielded Fires	N Gravestock
98/4	Predicting Ignition Time Under Transient Heat Flux Using Results from Constant Flux Experiments	A Henderson
98/5	Comparison Studies of Zone and CFD Fire Simulations	A Lovatt
98/6	Bench Scale Testing of Light Timber Frame Walls	P Olsson
98/7	Exploratory Salt Water Experiments of Balcony Spill Plume Using Laser Induced Fluorescence Technique	E Y Yii
99/1	Fire Safety and Security in Schools	R A Carter
99/2	A Review of the Building Separation Requirements of the New Zealand Building Code Acceptable Solutions	J M Clarke
99/3	Effect of Safety Factors in Timed Human Egress Simulations	K M Crawford
99/4	Fire Response of HVAC Systems in Multistorey Buildings: An Examination of the NZBC Acceptable Solutions	M Dixon
99/5	The Effectiveness of the Domestic Smoke Alarm Signal	C Duncan

99/6	Post-flashover Design Fires	R Feasey
99/7	An Analysis of Furniture Heat Release Rates by the Nordtest	J Firestone
99/8	Design for Escape from Fire	I J Garrett
99/9	Class A Foam Water Sprinkler Systems	D B Hipkins
99/10	Review of the New Zealand Standard for Concrete Structures (NZS 3101) for High Strength and Lightweight Concrete Exposed to Fire	M J Inwood
99/12	An Analytical Model for Vertical Flame Spread on Solids: An Initial Investigation	G A North
99/13	Should Bedroom Doors be Open or Closed While People are Sleeping? - A Probabilistic Risk Assessment	D L Palmer
99/14	Peoples Awareness of Fire	S J Rusbridge
99/15	Smoke Explosions	B J Sutherland
99/16	Reliability of Structural Fire Design	JKS Wong
00/1	Fire Spread on Exterior Walls	FNP Bong
00/2	Fire Resistance of Lightweight Framed Construction	PCR Collier
00/3	Fire Fighting Water: A Review of Fire Fighting Water Requirements (A New Zealand Perspective)	S Davis
00/4	The Combustion Behaviour of Upholstered Furniture Materials in New Zealand	H Denize
00/5	Full-Scale Compartment Fire Experiments on Upholstered Furniture	N Girgis
00/6	Fire Rated Seismic Joints	M James
00/7	Fire Design of Steel Members	K R Lewis
00/8	Stability of Precast Concrete Tilt Panels in Fire	L Lim
00/9	Heat Transfer Program for the Design of Structures Exposed to Fire	J Mason
00/10	An Analysis of Pre-Flashover Fire Experiments with Field Modelling Comparisons	C Nielsen
00/11	Fire Engineering Design Problems at Building Consent Stage	P Teo
00/12	A Comparison of Data Reduction Techniques for Zone Model Validation	S Weaver
00/13	Effect of Surface Area and Thickness on Fire Loads	H W Yii

School of Engineering
University of Canterbury
Private Bag 4800, Christchurch, New Zealand

Phone 643 364-2250
Fax 643 364-2758